Trademarks
Linux is a registered trademark of Linus Torvalds. PathScale is a registered trademark of Cray, Inc. Red Hat and all Red Hat-based trademarks are trademarks or registered trademarks of Red Hat, Inc. SUSE is a registered trademark of Novell, Inc. PGI is a registered trademark of NVIDIA Corporation. FLEXlm is a registered trademark of Flexera Software, Inc. PBS Professional, PBS Pro, and Green Provisioning are trademarks of Altair Engineering, Inc. All other trademarks are the property of their respective owners.

Rights and Restrictions
All statements, specifications, recommendations, and technical information contained herein are current or planned as of the date of publication of this document. They are reliable as of the time of this writing and are presented without warranty of any kind, expressed or implied. Bright Computing, Inc. shall not be liable for technical or editorial errors or omissions which may occur in this document. Bright Computing, Inc. shall not be liable for any damages resulting from the use of this document.

Limitation of Liability and Damages Pertaining to Bright Computing, Inc.
The Bright Cluster Manager product principally consists of free software that is licensed by the Linux authors free of charge. Bright Computing, Inc. shall have no liability nor will Bright Computing, Inc. provide any warranty for the Bright Cluster Manager to the extent that is permitted by law. Unless confirmed in writing, the Linux authors and/or third parties provide the program as is without any warranty, either expressed or implied, including, but not limited to, marketability or suitability for a specific purpose. The user of the Bright Cluster Manager product shall accept the full risk for the quality or performance of the product. Should the product malfunction, the costs for repair, service, or correction will be borne by the user of the Bright Cluster Manager product. No copyright owner or third party who has modified or distributed the program as permitted in this license shall be held liable for damages, including general or specific damages, damages caused by side effects or consequential damages, resulting from the use of the program or the un-usability of the program (including, but not limited to, loss of data, incorrect processing of data, losses that must be borne by you or others, or the inability of the program to work together with any other program), even if a copyright owner or third party had been advised about the possibility of such damages unless such copyright owner or third party has signed a writing to the contrary.
# Table of Contents

- Table of Contents .......................... i
- 0.1 Quickstart ................................ xv
- 0.2 About This Manual ..................... xv
- 0.3 About The Manuals In General .......... xv
- 0.4 Getting Administrator-Level Support .... xvi
- 0.5 Getting Professional Services ........ xvi

## 1 Introduction

1.1 Bright Cluster Manager Functions And Aims ........................................ 1
1.2 The Scope Of The Administrator Manual (This Manual) .......................... 1
  1.2.1 Installation ................................................ 1
  1.2.2 Configuration, Management, And Monitoring Via Bright Cluster Manager Tools And Applications .................. 2
1.3 Outside The Direct Scope Of The Administrator Manual ....................... 2

## 2 Cluster Management With Bright Cluster Manager

2.1 Concepts ........................................ 5
  2.1.1 Devices ............................................... 5
  2.1.2 Software Images ................................. 6
  2.1.3 Node Categories ......................... 7
  2.1.4 Node Groups ................................. 7
  2.1.5 Roles ............................................... 8
2.2 Modules Environment .................................. 8
  2.2.1 Adding And Removing Modules .... 8
  2.2.2 Using Local And Shared Modules .... 9
  2.2.3 Setting Up A Default Environment For All Users 9
  2.2.4 Creating A Modules Environment Module .... 10
  2.2.5 Lua Modules Environment (LMod) .... 10
2.3 Authentication ................................ 11
  2.3.1 Changing Administrative Passwords On The Cluster .... 11
  2.3.2 Logins Using ssh ...................... 12
  2.3.3 Certificates .................................. 13
  2.3.4 Profiles .................................. 14
2.4 Bright View GUI ................................. 14
  2.4.1 Installing The Cluster Management GUI Service .... 14
  2.4.2 Navigating The Cluster With Bright View .... 15
2.5 Cluster Management Shell .................. 18
  2.5.1 Invoking cmsh ................................. 18
  2.5.2 Levels, Modes, Help, And Commands Syntax In cmsh .... 20
  2.5.3 Working With Objects ................. 24
  2.5.4 Accessing Cluster Settings ........... 33
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.5.5</td>
<td>Advanced cmsh Features</td>
<td>34</td>
</tr>
<tr>
<td>2.6</td>
<td>Cluster Management Daemon</td>
<td>44</td>
</tr>
<tr>
<td>2.6.1</td>
<td>Controlling The Cluster Management Daemon</td>
<td>45</td>
</tr>
<tr>
<td>2.6.2</td>
<td>Configuring The Cluster Management Daemon</td>
<td>46</td>
</tr>
<tr>
<td>2.6.3</td>
<td>CMDaemon Versions</td>
<td>46</td>
</tr>
<tr>
<td>2.6.4</td>
<td>Configuring The Cluster Management Daemon Logging Facilities</td>
<td>47</td>
</tr>
<tr>
<td>2.6.5</td>
<td>Configuration File Modification, And The FrozenFile Directive</td>
<td>48</td>
</tr>
<tr>
<td>2.6.6</td>
<td>Configuration File Conflicts Between The Standard Distribution And Bright Cluster Manager For Generated And Non-Generated Files</td>
<td>49</td>
</tr>
<tr>
<td>2.6.7</td>
<td>CMDaemon Lite</td>
<td>49</td>
</tr>
<tr>
<td>3</td>
<td>Configuring The Cluster</td>
<td>53</td>
</tr>
<tr>
<td>3.1</td>
<td>Main Cluster Configuration Settings</td>
<td>54</td>
</tr>
<tr>
<td>3.1.1</td>
<td>Cluster Configuration: Various Name-Related Settings</td>
<td>54</td>
</tr>
<tr>
<td>3.1.2</td>
<td>Cluster Configuration: Some Network-Related Settings</td>
<td>55</td>
</tr>
<tr>
<td>3.1.3</td>
<td>Miscellaneous Settings</td>
<td>56</td>
</tr>
<tr>
<td>3.1.4</td>
<td>Limiting The Maximum Number Of Open Files</td>
<td>58</td>
</tr>
<tr>
<td>3.2</td>
<td>Network Settings</td>
<td>59</td>
</tr>
<tr>
<td>3.2.1</td>
<td>Configuring Networks</td>
<td>60</td>
</tr>
<tr>
<td>3.2.2</td>
<td>Adding Networks</td>
<td>64</td>
</tr>
<tr>
<td>3.2.3</td>
<td>Changing Network Parameters</td>
<td>64</td>
</tr>
<tr>
<td>3.2.4</td>
<td>Tools For Viewing Cluster Connections</td>
<td>75</td>
</tr>
<tr>
<td>3.3</td>
<td>Configuring Bridge Interfaces</td>
<td>78</td>
</tr>
<tr>
<td>3.4</td>
<td>Configuring VLAN interfaces</td>
<td>79</td>
</tr>
<tr>
<td>3.4.1</td>
<td>Configuring A VLAN Interface Using cmsh</td>
<td>79</td>
</tr>
<tr>
<td>3.4.2</td>
<td>Configuring A VLAN Interface Using Bright View</td>
<td>80</td>
</tr>
<tr>
<td>3.5</td>
<td>Configuring Bonded Interfaces</td>
<td>80</td>
</tr>
<tr>
<td>3.5.1</td>
<td>Adding A Bonded Interface</td>
<td>80</td>
</tr>
<tr>
<td>3.5.2</td>
<td>Single Bonded Interface On A Regular Node</td>
<td>81</td>
</tr>
<tr>
<td>3.5.3</td>
<td>Multiple Bonded Interface On A Regular Node</td>
<td>81</td>
</tr>
<tr>
<td>3.5.4</td>
<td>Bonded Interfaces On Head Nodes And HA Head Nodes</td>
<td>82</td>
</tr>
<tr>
<td>3.5.5</td>
<td>Tagged VLAN On Top Of A Bonded Interface</td>
<td>82</td>
</tr>
<tr>
<td>3.5.6</td>
<td>Further Notes On Bonding</td>
<td>82</td>
</tr>
<tr>
<td>3.6</td>
<td>Configuring InfiniBand And Omni-Path Interfaces</td>
<td>83</td>
</tr>
<tr>
<td>3.6.1</td>
<td>Installing Software Packages</td>
<td>83</td>
</tr>
<tr>
<td>3.6.2</td>
<td>Subnet Managers</td>
<td>83</td>
</tr>
<tr>
<td>3.6.3</td>
<td>InfiniBand Network Settings</td>
<td>84</td>
</tr>
<tr>
<td>3.6.4</td>
<td>Verifying Connectivity</td>
<td>85</td>
</tr>
<tr>
<td>3.7</td>
<td>Configuring BMC (IPMI/iLO/DRAC/CIMC/Redfish) Interfaces</td>
<td>86</td>
</tr>
<tr>
<td>3.7.1</td>
<td>BMC Network Settings</td>
<td>87</td>
</tr>
<tr>
<td>3.7.2</td>
<td>BMC Authentication</td>
<td>88</td>
</tr>
<tr>
<td>3.7.3</td>
<td>Interfaces Settings</td>
<td>89</td>
</tr>
<tr>
<td>3.7.4</td>
<td>Identification With A BMC</td>
<td>90</td>
</tr>
<tr>
<td>3.8</td>
<td>Configuring Switches And PDUs</td>
<td>90</td>
</tr>
<tr>
<td>3.8.1</td>
<td>Configuring With The Manufacturer’s Configuration Interface</td>
<td>90</td>
</tr>
<tr>
<td>3.8.2</td>
<td>Configuring SNMP</td>
<td>91</td>
</tr>
<tr>
<td>3.8.3</td>
<td>Uplink Ports</td>
<td>91</td>
</tr>
<tr>
<td>Section</td>
<td>Page</td>
<td></td>
</tr>
<tr>
<td>------------------------------------------------------------------------</td>
<td>------</td>
<td></td>
</tr>
<tr>
<td>3.8.4 The <code>showport</code> MAC Address to Port Matching Tool</td>
<td>92</td>
<td></td>
</tr>
<tr>
<td>3.8.5 Disabling Port Detection</td>
<td>93</td>
<td></td>
</tr>
<tr>
<td>3.8.6 The <code>switchoverview</code> Command</td>
<td>93</td>
<td></td>
</tr>
<tr>
<td>3.9 Disk Layouts: Disked, Semi-Diskless, And Diskless Node Configuration</td>
<td>94</td>
<td></td>
</tr>
<tr>
<td>3.9.1 Disk Layouts</td>
<td>94</td>
<td></td>
</tr>
<tr>
<td>3.9.2 Disk Layout Assertions</td>
<td>94</td>
<td></td>
</tr>
<tr>
<td>3.9.3 Changing Disk Layouts</td>
<td>95</td>
<td></td>
</tr>
<tr>
<td>3.9.4 Changing A Disk Layout From Disked To Diskless</td>
<td>95</td>
<td></td>
</tr>
<tr>
<td>3.10 Configuring NFS Volume Exports And Mounts</td>
<td>96</td>
<td></td>
</tr>
<tr>
<td>3.10.1 Exporting A Filesystem Using Bright View And <code>cmsh</code></td>
<td>97</td>
<td></td>
</tr>
<tr>
<td>3.10.2 Mounting A Filesystem Using Bright View And <code>cmsh</code></td>
<td>99</td>
<td></td>
</tr>
<tr>
<td>3.10.3 Mounting A Filesystem Subtree For A Diskless Node Over NFS</td>
<td>102</td>
<td></td>
</tr>
<tr>
<td>3.10.4 Mounting The Root Filesystem For A Diskless Node Over NFS</td>
<td>104</td>
<td></td>
</tr>
<tr>
<td>3.10.5 Configuring NFS Volume Exports And Mounts Over RDMA With OFED Drivers</td>
<td>106</td>
<td></td>
</tr>
<tr>
<td>3.11 Managing And Configuring Services</td>
<td>107</td>
<td></td>
</tr>
<tr>
<td>3.11.1 Why Use The Cluster Manager For Services?</td>
<td>107</td>
<td></td>
</tr>
<tr>
<td>3.11.2 Managing And Configuring Services—Examples</td>
<td>108</td>
<td></td>
</tr>
<tr>
<td>3.12 Managing And Configuring A Rack</td>
<td>111</td>
<td></td>
</tr>
<tr>
<td>3.12.1 Racks</td>
<td>111</td>
<td></td>
</tr>
<tr>
<td>3.12.2 Assigning Devices To A Rack</td>
<td>113</td>
<td></td>
</tr>
<tr>
<td>3.12.3 Assigning Devices To A Chassis</td>
<td>114</td>
<td></td>
</tr>
<tr>
<td>3.13 Configuring A GPU Unit, And Configuring GPU Settings</td>
<td>117</td>
<td></td>
</tr>
<tr>
<td>3.13.1 GPUs And GPU Units</td>
<td>117</td>
<td></td>
</tr>
<tr>
<td>3.13.2 GPU Unit Configuration Example: The Dell PowerEdge C410x</td>
<td>117</td>
<td></td>
</tr>
<tr>
<td>3.13.3 Configuring GPU Settings</td>
<td>119</td>
<td></td>
</tr>
<tr>
<td>3.14 Configuring Custom Scripts</td>
<td>126</td>
<td></td>
</tr>
<tr>
<td>3.14.1 <code>custompowerscript</code></td>
<td>126</td>
<td></td>
</tr>
<tr>
<td>3.14.2 <code>custompingscript</code></td>
<td>126</td>
<td></td>
</tr>
<tr>
<td>3.14.3 <code>customremoteconsolescript</code></td>
<td>126</td>
<td></td>
</tr>
<tr>
<td>3.15 Cluster Configuration Without Execution By CMDaemon</td>
<td>127</td>
<td></td>
</tr>
<tr>
<td>3.15.1 Cluster Configuration: The Bigger Picture</td>
<td>127</td>
<td></td>
</tr>
<tr>
<td>3.15.2 Making Nodes Function Differently By Image</td>
<td>128</td>
<td></td>
</tr>
<tr>
<td>3.15.3 Making All Nodes Function Differently From Normal Cluster Behavior With FrozenFile</td>
<td>130</td>
<td></td>
</tr>
<tr>
<td>3.15.4 Adding Functionality To Nodes Via An <code>initialize</code> Or <code>finalize</code> Script</td>
<td>131</td>
<td></td>
</tr>
<tr>
<td>3.15.5 Examples Of Configuring Nodes With Or Without CMDaemon</td>
<td>131</td>
<td></td>
</tr>
<tr>
<td>4 Power Management</td>
<td>133</td>
<td></td>
</tr>
<tr>
<td>4.1 Configuring Power Parameters</td>
<td>133</td>
<td></td>
</tr>
<tr>
<td>4.1.1 PDU-Based Power Control</td>
<td>134</td>
<td></td>
</tr>
<tr>
<td>4.1.2 IPMI-Based Power Control</td>
<td>135</td>
<td></td>
</tr>
<tr>
<td>4.1.3 Combining PDU- and IPMI-Based Power Control</td>
<td>136</td>
<td></td>
</tr>
<tr>
<td>4.1.4 Custom Power Control</td>
<td>136</td>
<td></td>
</tr>
<tr>
<td>4.1.5 Hewlett Packard iLO-Based Power Control</td>
<td>137</td>
<td></td>
</tr>
<tr>
<td>4.1.6 Dell <code>drac</code>-based Power Control</td>
<td>138</td>
<td></td>
</tr>
<tr>
<td>4.1.7 Redfish-Based and CIMC-Based Power Control</td>
<td>138</td>
<td></td>
</tr>
<tr>
<td>4.2 Power Operations</td>
<td>138</td>
<td></td>
</tr>
</tbody>
</table>
4.2.1 Power Operations Overview ........................................ 138
4.2.2 Power Operations With Bright View ................................. 139
4.2.3 Power Operations Through cmsh ................................. 139
4.3 Monitoring Power .................................................. 143
4.4 CPU Scaling Governors .............................................. 143
4.4.1 The Linux Kernel And CPU Scaling Governors ..................... 143
4.4.2 The Governor List According To sysinfo ........................... 143
4.4.3 Setting The Governor ............................................ 144
4.5 Switch Configuration To Survive Power Downs ...................... 145

5 Node Provisioning ......................................................... 147
5.1 Before The Kernel Loads ............................................. 147
5.1.1 PXE Booting .................................................... 147
5.1.2 iPXE Booting From A Disk Drive .................................. 151
5.1.3 iPXE Booting Using InfiniBand ................................... 151
5.1.4 Using PXE To Boot From The Drive ................................. 152
5.1.5 Network Booting Without PXE On The ARMv8 Architecture ....... 152
5.1.6 The Boot Role .................................................... 152
5.2 Provisioning Nodes .................................................. 152
5.2.1 Provisioning Nodes: Configuration Settings ......................... 152
5.2.2 Provisioning Nodes: Role Setup With cmsh ......................... 153
5.2.3 Provisioning Nodes: Role Setup With Bright View .................. 154
5.2.4 Provisioning Nodes: Housekeeping ................................ 156
5.3 The Kernel Image, Ramdisk And Kernel Modules ..................... 159
5.3.1 Booting To A “Good State” Software Image ......................... 159
5.3.2 Selecting Kernel Driver Modules To Load Onto Nodes ............... 159
5.3.3 InfiniBand Provisioning ........................................ 161
5.3.4 Omni-Path Provisioning ........................................ 163
5.3.5 VLAN Provisioning ............................................. 163
5.4 Node-Installer ....................................................... 164
5.4.1 Requesting A Node Certificate ................................... 164
5.4.2 Deciding Or Selecting Node Configuration ......................... 166
5.4.3 Starting Up All Network Interfaces ................................ 176
5.4.4 Determining Install-mode Type And Execution Mode ............... 178
5.4.5 Running Initialize Scripts ....................................... 183
5.4.6 Checking Partitions, RAID Configuration, Mounting Filesystems .... 183
5.4.7 Synchronizing The Local Drive With The Software Image ........... 184
5.4.8 Writing Network Configuration Files ................................ 187
5.4.9 Creating A Local /etc/fstab File .................................. 188
5.4.10 Installing GRUB Bootloader ...................................... 188
5.4.11 Running Finalize Scripts ....................................... 190
5.4.12 Unloading Specific Drivers ..................................... 190
5.4.13 Switching To The Local init Process .............................. 190
5.5 Node States .......................................................... 190
5.5.1 Node States Icons In Bright View ................................ 191
5.5.2 Node States Shown In cmsh ....................................... 191
5.5.3 Node States Indicating Regular Start Up .......................... 191
# Table of Contents

5.5.4 Node States That May Indicate Problems .............................................. 192
5.6 Updating Running Nodes ........................................................................... 194
   5.6.1 Updating Running Nodes: Configuration With \texttt{excludelistupdate} ................................................. 194
   5.6.2 Updating Running Nodes: With \texttt{cmsh} Using \texttt{imageupdate} ...................................................... 202
   5.6.3 Updating Running Nodes: With Bright View Using the \texttt{update node} Option ............................. 202
   5.6.4 Updating Running Nodes: Considerations .......................................................... 202
5.7 Adding New Nodes ..................................................................................... 203
   5.7.1 Adding New Nodes With \texttt{cmsh} And Bright View Add Functions ................................. 203
   5.7.2 Adding New Nodes With The Node Creation Wizard ........................................ 203
5.8 Troubleshooting The Node Boot Process .................................................... 204
   5.8.1 Node Fails To PXE Boot .......................................................................... 205
   5.8.2 Node-installer Logging ............................................................................. 207
   5.8.3 Provisioning Logging ................................................................................ 208
   5.8.4 Ramdisk Fails During Loading Or Sometime Later ............................................ 209
   5.8.5 Ramdisk Cannot Start Network ..................................................................... 209
   5.8.6 Node-Installer Cannot Create Disk Layout ...................................................... 210
   5.8.7 Node-Installer Cannot Start BMC (IPMI/iLO) Interface ..................................... 213

6 User Management .......................................................................................... 217
   6.1 Managing Users And Groups With Bright View ........................................... 217
   6.2 Managing Users And Groups With \texttt{cmsh} ..................................................... 219
      6.2.1 Adding A User ......................................................................................... 219
      6.2.2 Saving The Modified State ....................................................................... 220
      6.2.3 Editing Properties Of Users And Groups .................................................... 221
      6.2.4 Reverting To The Unmodified State ............................................................ 223
      6.2.5 Removing A User ..................................................................................... 224
   6.3 Using An External LDAP Server .................................................................. 224
      6.3.1 External LDAP Server Replication .............................................................. 226
      6.3.2 High Availability ....................................................................................... 228
   6.4 Tokens And Profiles .................................................................................... 229
      6.4.1 Modifying Profiles .................................................................................... 231
      6.4.2 Creation Of Custom Certificates With Profiles, For Users Managed By Bright Cluster Manager’s Internal LDAP .......................................................... 231
      6.4.3 Creation Of Custom Certificates With Profiles, For Users Managed By An External LDAP ........................................................................................................ 234
      6.4.4 Logging The Actions Of CMDaemon Users .................................................. 235

7 Workload Management .................................................................................... 237
   7.1 Workload Managers Choices ....................................................................... 237
   7.2 Forcing Jobs To Run In A Workload Management System ............................ 238
      7.2.1 Disallowing User Logins To Regular Nodes Via \texttt{cmsh} ........................................ 238
      7.2.2 Disallowing User Logins To Regular Nodes Via Bright View ....................... 239
      7.2.3 Disallowing Other User Processes Outside Of Workload Manager User Processes 239
   7.3 Installation Of Workload Managers ............................................................... 239
      7.3.1 Setting Up, Enabling, And Disabling The Workload Manager With \texttt{wlm-setup} 240
      7.3.2 Other Options With \texttt{wlm-setup} .................................................................... 241
      7.3.3 Prolog And Epilog Scripts .......................................................................... 243
## Table of Contents

7.4 Enabling, Disabling, And Monitoring Workload Managers ........................................ 245
7.4.1 Enabling And Disabling A Workload Manager With Bright View .......................... 246
7.4.2 Enabling And Disabling A Workload Manager With cmsh .................................. 248
7.4.3 Monitoring The Workload Manager Services ....................................................... 250
7.5 Configuring And Running Individual Workload Managers ....................................... 251
7.5.1 Configuring And Running Slurm ........................................................................... 251
7.5.2 Configuring And Running SGE ............................................................................. 256
7.5.3 Installing, Configuring, And Running UGE ......................................................... 258
7.5.4 Configuring And Running PBS Pro ....................................................................... 264
7.5.5 Installing, Configuring, And Running LSF ........................................................... 268
7.6 Using Bright View With Workload Management ..................................................... 274
7.6.1 Jobs Display And Handling In Bright View ......................................................... 274
7.6.2 Queues Display And Handling In Bright View ..................................................... 274
7.7 Using cmsh With Workload Management .............................................................. 275
7.7.1 Jobs Display And Handling In cmsh: jobs Mode ................................................ 275
7.7.2 Job Queues Display And Handling In cmsh: jobqueue Mode .............................. 279
7.7.3 Nodes Drainage Status And Handling In cmsh .................................................... 281
7.7.4 Launching Jobs With cm-launcher ....................................................................... 283
7.8 Examples Of Workload Management Assignment .................................................. 284
7.8.1 Setting Up A New Category And A New Queue For It .......................................... 284
7.8.2 Setting Up A Prejob Check .................................................................................. 287
7.9 Power Saving Features ............................................................................................. 288
7.9.1 Slurm .................................................................................................................. 288
7.9.2 The cm-scale Service ......................................................................................... 289
7.10 Cgroups ................................................................................................................... 289
7.10.1 Cgroups Settings For Workload Managers ....................................................... 289
7.10.2 Managed Cgroups ............................................................................................ 294
7.11 Custom Node Parameters ....................................................................................... 298

8 The cm-scale Service .................................................................................................. 301
8.1 Introduction .............................................................................................................. 301
8.1.1 Use Cases ............................................................................................................ 301
8.1.2 Restrictions ......................................................................................................... 302
8.1.3 Setup ................................................................................................................... 303
8.1.4 Comparison With cm-scale-cluster .................................................................... 303
8.2 Configuration ........................................................................................................... 303
8.2.1 The ScaleServer Role ....................................................................................... 303
8.2.2 Resource Providers ............................................................................................. 305
8.2.3 Time Quanta Optimization ................................................................................ 307
8.2.4 Fairsharing Priority Calculations And Node Management ............................... 310
8.2.5 Engines ............................................................................................................... 310
8.2.6 Trackers .............................................................................................................. 311
8.3 Examples Of cm-scale Use ..................................................................................... 316
8.3.1 Simple Static Node Provider Usage Example ..................................................... 316
8.3.2 Simple Dynamic Node Provider Usage Example .............................................. 319
8.4 Further cm-scale Configuration And Examples ..................................................... 325
8.4.1 Dynamic Nodes Repurposing ............................................................................ 325
# Table of Contents

## 9.9.1 Updating CMDaemon

## 9.9.2 Generating The Kubernetes Setup File `cm-kubernetes-setup.conf`

## 9.9.3 Secondary IP Address Configuration

## 9.9.4 Installing Azure CNI

## 9.9.5 Installing Azure NPM

## 9.9.6 Ensuring Some Firewall Rules Are Applied Via Kubelet Service

## 9.9.7 DNS Fix For Ubuntu Installations

## 9.9.8 Restarting Services

## 10 BeeGFS

### 10.1 BeeGFS Introduction

### 10.1.1 BeeGFS Concepts

### 10.1.2 BeeGFS Installation Notes And Options

### 10.2 Deployment And Uninstallation Of BeeGFS With `cm-beegfs-setup`

#### 10.2.1 Deployment Of BeeGFS

#### 10.2.2 Uninstalling BeeGFS

### 10.3 Managing The Deployed BeeGFS Instance

#### 10.3.1 Setup

#### 10.3.2 BeeGFS Objects

#### 10.3.3 Usage

#### 10.3.4 Admon Interface

## 11 Post-Installation Software Management

### 11.1 Bright Cluster Manager Packages And Their Naming Convention

### 11.2 Managing Packages On The Head Node

#### 11.2.1 Managing RPM Or `.deb` Packages On The Head Node

#### 11.2.2 Installation Of Packages On The Head Node That Are Not `.deb` And Not `.rpm`

### 11.3 Kernel Management On A Head Node Or Image

#### 11.3.1 Installing A Standard Distribution Kernel Into An Image Or On A Head Node

#### 11.3.2 Excluding Kernels And Other Packages From Updates

#### 11.3.3 Updating A Kernel In A Software Image

#### 11.3.4 Setting Kernel Options For Software Images

#### 11.3.5 Kernel Driver Modules

### 11.4 Managing A Package In A Software Image And Running It On Nodes

#### 11.4.1 Installing From Head Into The Image: Changing The Root Directory Into Which The Packages Are Deployed

#### 11.4.2 Installing From Head Into The Image: Updating The Node

#### 11.4.3 Installing From Head Into The Image: Possible Issues When Using `rpm --root`, `yum --installroot` Or `chroot`

### 11.5 Managing Non-RPM Software In A Software Image And Running It On Nodes

### 11.6 Creating A Custom Software Image

#### 11.6.1 Creating A Base Distribution Archive From A Base Host

#### 11.6.2 Creating The Software Image With `cm-create-image`
11.6.3 Configuring Local Repositories For Linux Distributions, And For The Bright Cluster Manager Package Repository, For A Software Image ........................................... 453
11.6.4 Creating A Custom Image From The Local Repository ........................................... 456

12 Cluster Monitoring ........................................... 457
12.1 A Basic Monitoring Example And Action ........................................... 457
12.1.1 Synopsis Of Basic Monitoring Example ........................................... 457
12.1.2 Before Using The Basic Monitoring Example—Setting Up The Pieces 458
12.1.3 Using The Monitoring Basic Example ........................................... 459
12.2 Monitoring Concepts And Definitions ........................................... 462
12.2.1 Measurables ........................................... 462
12.2.2 Enummetrics ........................................... 464
12.2.3 Metrics ........................................... 465
12.2.4 Health Check ........................................... 466
12.2.5 Trigger ........................................... 468
12.2.6 Action ........................................... 468
12.2.7 Severity ........................................... 469
12.2.8 AlertLevel ........................................... 469
12.2.9 Flapping ........................................... 469
12.2.10 Data Producer ........................................... 469
12.2.11 Conceptual Overview: The Main Monitoring Interfaces Of Bright View 473
12.3 Monitoring Visualization With Bright View ........................................... 474
12.3.1 The Monitoring Window ........................................... 474
12.4 Monitoring Configuration With Bright View ........................................... 476
12.4.1 Monitoring Configuration: Data Producers ........................................... 476
12.4.2 Monitoring Configuration: Measurables ........................................... 478
12.4.3 Monitoring Configuration: Consolidators ........................................... 480
12.4.4 Monitoring Configuration: Actions ........................................... 483
12.4.5 Monitoring Configuration: Triggers ........................................... 485
12.4.6 Monitoring Configuration: Health status ........................................... 488
12.4.7 Monitoring Configuration: All Health Checks ........................................... 489
12.4.8 Monitoring Configuration: Standalone Monitored Entities ........................................... 490
12.5 The monitoring Mode Of cmsh ........................................... 490
12.5.1 The action Submode ........................................... 491
12.5.2 The consolidator Submode ........................................... 493
12.5.3 The measurable Submode ........................................... 495
12.5.4 The setup Submode ........................................... 499
12.5.5 The standalone Submode ........................................... 505
12.5.6 The trigger Submode ........................................... 505
12.6 Obtaining Monitoring Data Values ........................................... 508
12.6.1 Getting The List Of Measurables For An Entity: The measurable, metrics, healthchecks And enummetrics Commands ........................................... 508
12.6.2 On-Demand Metric Sampling And Health Checks ........................................... 509
12.6.3 The Latest Data And Counter Values—The latest*data And latestmetriccounters Commands ........................................... 511
12.6.4 Data Values Over A Period—The dumpmonitoringdata Command ........................................... 513
12.6.5 Monitoring Data Health Overview—The healthoverview Command ........................................... 519
### Table of Contents

12.6.6 Monitoring Data About The Monitoring System—The `monitoringinfo` Command 520
12.6.7 Dropping Monitoring Data With The `monitoringdrop` Command .................. 521
12.6.8 Monitoring Suspension And Resumption—The `monitoringsuspend` And `monitoringresume` Commands ................................. 522

12.7 The User Portal .......................................................... 523
12.7.1 Accessing The User Portal ........................................ 523
12.7.2 Setting A Common Username/Password For The User Portal ..................... 524
12.7.3 User Portal Home Page ............................................. 525

12.8 Job Monitoring .............................................................. 526
12.8.1 Job Metrics Introduction ........................................... 526
12.8.2 Job Information Retention .......................................... 527
12.8.3 Job Metrics Sampling Configuration ................................ 528
12.8.4 Job Monitoring In `cmsh` ........................................... 529

12.9 Job Accounting ............................................................... 530
12.9.1 Labeled Entities ...................................................... 531
12.9.2 PromQL Queries ....................................................... 532
12.9.3 Job Accounting In Bright View .................................... 537
12.9.4 PromQL Query Modes And Specification In Bright View ...................... 540

12.10 Event Viewer ................................................................. 542
12.10.1 Viewing Events In Bright View .................................... 542
12.10.2 Viewing Events In `cmsh` ........................................... 542
12.10.3 Using The Event Bucket From The Shell For Events And For Tagging Device States 544
12.10.4 InfoMessages .......................................................... 545

13 Day-to-day Administration .................................................... 547
13.1 Parallel Shells: `pdsh` And `pexec` .................................... 547
13.1.1 `pdsh` In The OS Shell ............................................... 547
13.1.2 `pexec` In `cmsh` ..................................................... 551
13.1.3 `pexec` In Bright View .............................................. 551
13.1.4 Using The `-j|--join` Option Of `pexec` In `cmsh` ....................... 552
13.1.5 Other Parallel Commands ............................................ 552
13.2 Getting Support With Cluster Manager Issues .................................... 553
13.2.1 Support Via E-mail .................................................. 553
13.2.2 Reporting Cluster Manager Diagnostics With `cm-diagnose` ............... 554
13.2.3 Requesting Remote Support With `request-remote-assistance` .......... 555
13.2.4 Requesting Remote Support With A Shared Screen Utility ............... 556

13.3 Backups ................................................................. 557
13.3.1 Cluster Installation Backup ....................................... 557
13.3.2 Local Database And Data Backups And Restoration ....................... 557

13.4 Revision Control For Images ........................................... 559
13.4.1 Btrfs: The Concept And Why It Works Well In Revision Control For Images 559
13.4.2 Btrfs Availability And Distribution Support ................................ 560
13.4.3 Installing Btrfs To Work With Revision Control Of Images In Bright Cluster Manager 560
13.4.4 Using `cmsh` For Revision Control Of Images .......................... 562

13.5 BIOS Configuration And Updates ....................................... 565
13.5.1 Updating BIOS ....................................................... 565
13.5.2 Booting DOS Image .................................................. 565
# Table of Contents

13.6 Hardware Match Check With The `hardware-profile` Data Producer ................................ 566
13.7 Serial Over LAN Console Access ................................................................. 567
   13.7.1 Background Notes On Serial Console And SOL .......................................... 567
   13.7.2 SOL Console Configuration With Bright View ............................................. 569
   13.7.3 SOL Console Configuration And Access With `cmsh` ...................................... 569
   13.7.4 The `conman` Serial Console Logger And Viewer ......................................... 570
13.8 Managing Raw Monitoring Data ................................................................. 574
   13.8.1 Monitoring Subsystem Disk Usage With The `monitoringinfo --storage` Option ................ 574
   13.8.2 Estimating The Required Size Of The Storage Device .................................... 574
   13.8.3 Moving Monitoring Data Elsewhere .......................................................... 575
   13.8.4 Reducing Monitoring Data By Reducing Samples ......................................... 575
   13.8.5 Deleting All Monitoring Data .................................................................. 575
13.9 Node Replacement ......................................................................................... 576

14 High Availability ......................................................................................... 579
14.0 Introduction ............................................................................................... 579
   14.0.1 Why Have High Availability? ............................................................... 579
   14.0.2 High Availability Is Possible On Head Nodes, And Also On Regular Nodes .......... 579
   14.0.3 High Availability Usually Uses Shared Storage ............................................. 579
   14.0.4 Organization Of This Chapter ............................................................... 579
14.1 HA Concepts .............................................................................................. 580
   14.1.1 Primary, Secondary, Active, Passive .......................................................... 580
   14.1.2 Monitoring The Active Head Node, Initiating Failover .................................. 580
   14.1.3 Services In Bright Cluster Manager HA Setups ............................................. 580
   14.1.4 Failover Network Topology ................................................................... 581
   14.1.5 Shared Storage ..................................................................................... 583
   14.1.6 Guaranteeing One Active Head At All Times ............................................ 584
   14.1.7 Automatic Vs Manual Failover ............................................................... 585
   14.1.8 HA And Cloud Nodes ............................................................................ 586
   14.1.9 HA Using Virtual Head Nodes ............................................................... 586
14.2 HA Setup Procedure Using `cmha-setup` .................................................. 586
   14.2.1 Preparation ......................................................................................... 587
   14.2.2 Failover Cloning .................................................................................. 589
   14.2.3 Shared Storage Setup ............................................................................ 592
   14.2.4 Automated Failover And Relevant Testing .............................................. 594
14.3 Running `cmha-setup` Without Ncurses, Using An XML Specification ............... 595
   14.3.1 Why Run It Without Ncurses? ................................................................ 595
   14.3.2 The Syntax Of `cmha-setup` Without Ncurses ........................................... 595
   14.3.3 Example `cmha-setup` Run Without Ncurses .......................................... 596
14.4 Managing HA ............................................................................................. 596
   14.4.1 Changing An Existing Failover Configuration ........................................... 597
   14.4.2 `cmha` Utility .................................................................................... 597
   14.4.3 States .................................................................................................. 600
   14.4.4 Failover Action Decisions ................................................................... 601
   14.4.5 Keeping Head Nodes In Sync ................................................................ 602
   14.4.6 High Availability Parameters .............................................................. 604
   14.4.7 Viewing Failover Via Bright View .......................................................... 606
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>14.4.8</td>
<td>Re-cloning A Head Node</td>
<td>606</td>
</tr>
<tr>
<td>14.5</td>
<td>HA For Regular Nodes</td>
<td>607</td>
</tr>
<tr>
<td>14.5.1</td>
<td>Why Have HA On Regular Nodes?</td>
<td>607</td>
</tr>
<tr>
<td>14.5.2</td>
<td>Comparing Head And Regular Node HA</td>
<td>607</td>
</tr>
<tr>
<td>14.5.3</td>
<td>Setting Up A Regular Node HA Service</td>
<td>608</td>
</tr>
<tr>
<td>14.5.4</td>
<td>The Sequence Of Events When Making Another HA Regular Node Active</td>
<td>612</td>
</tr>
<tr>
<td>14.6</td>
<td>HA And Workload Manager Jobs</td>
<td>612</td>
</tr>
<tr>
<td>15</td>
<td>Dell BIOS Management</td>
<td>615</td>
</tr>
<tr>
<td>15.1</td>
<td>Introduction</td>
<td>615</td>
</tr>
<tr>
<td>15.2</td>
<td>Prerequisites For BIOS Management</td>
<td>615</td>
</tr>
<tr>
<td>15.3</td>
<td>BIOS settings</td>
<td>616</td>
</tr>
<tr>
<td>15.3.1</td>
<td>Initializing The BIOS Settings Via cmsh</td>
<td>616</td>
</tr>
<tr>
<td>15.3.2</td>
<td>Managing The BIOS Settings Via Bright View</td>
<td>616</td>
</tr>
<tr>
<td>15.3.3</td>
<td>Applying Dell Settings And Firmware Updates Via Bright View</td>
<td>619</td>
</tr>
<tr>
<td>15.3.4</td>
<td>Managing The BIOS Settings Via cmsh</td>
<td>619</td>
</tr>
<tr>
<td>15.4</td>
<td>Frequently Asked Questions</td>
<td>622</td>
</tr>
<tr>
<td>A</td>
<td>Generated Files</td>
<td>625</td>
</tr>
<tr>
<td>A.1</td>
<td>System Configuration Files Created Or Modified By CMDaemon On Head Nodes</td>
<td>625</td>
</tr>
<tr>
<td>A.2</td>
<td>System Configuration Files Created Or Modified By CMDaemon In Software Images:</td>
<td>627</td>
</tr>
<tr>
<td>A.3</td>
<td>Files Created On Regular Nodes By The Node-Installer</td>
<td>628</td>
</tr>
<tr>
<td>A.4</td>
<td>Files Not Generated, But Installed.</td>
<td>629</td>
</tr>
<tr>
<td>B</td>
<td>Bright Computing Public Key</td>
<td>633</td>
</tr>
<tr>
<td>C</td>
<td>CMDaemon Configuration File Directives</td>
<td>635</td>
</tr>
<tr>
<td>D</td>
<td>Disk Partitioning And RAID Configuration</td>
<td>663</td>
</tr>
<tr>
<td>D.1</td>
<td>Structure Of Partitioning Definition—The Global Partitioning XML Schema File</td>
<td>663</td>
</tr>
<tr>
<td>D.2</td>
<td>Structure Of Hardware RAID Definition—The Hardware RAID XML Schema File</td>
<td>670</td>
</tr>
<tr>
<td>D.3</td>
<td>Example: Default Node Partitioning</td>
<td>673</td>
</tr>
<tr>
<td>D.4</td>
<td>Example: Hardware RAID Configuration</td>
<td>674</td>
</tr>
<tr>
<td>D.4.1</td>
<td>RAID level 0 And RAID 10 Example</td>
<td>674</td>
</tr>
<tr>
<td>D.5</td>
<td>Example: Software RAID</td>
<td>676</td>
</tr>
<tr>
<td>D.6</td>
<td>Example: Software RAID With Swap</td>
<td>677</td>
</tr>
<tr>
<td>D.7</td>
<td>Example: Logical Volume Manager</td>
<td>678</td>
</tr>
<tr>
<td>D.8</td>
<td>Example: Logical Volume Manager With RAID 1</td>
<td>679</td>
</tr>
<tr>
<td>D.9</td>
<td>Example: Diskless</td>
<td>680</td>
</tr>
<tr>
<td>D.10</td>
<td>Example: Semi-diskless</td>
<td>681</td>
</tr>
<tr>
<td>D.11</td>
<td>Example: Preventing Accidental Data Loss</td>
<td>682</td>
</tr>
<tr>
<td>D.12</td>
<td>Example: Using Custom Assertions</td>
<td>683</td>
</tr>
<tr>
<td>E</td>
<td>Example initialize And finalize Scripts</td>
<td>685</td>
</tr>
<tr>
<td>E.1</td>
<td>When Are They Used?</td>
<td>685</td>
</tr>
<tr>
<td>E.2</td>
<td>Accessing From Bright View And cmsh</td>
<td>685</td>
</tr>
<tr>
<td>E.3</td>
<td>Environment Variables Available To initialize And finalize Scripts</td>
<td>686</td>
</tr>
<tr>
<td>E.4</td>
<td>Using Environment Variables Stored In Multiple Variables</td>
<td>688</td>
</tr>
</tbody>
</table>
## Table of Contents

E.5 Storing A Configuration To A Filesystem ............................... 689  
E.5.1 Storing With Initialize Scripts .................................. 689  
E.5.2 Ways Of Writing A Finalize Script To Configure The Destination Nodes 689  
E.5.3 Restricting The Script To Nodes Or Node Categories .......... 692  

F Workload Managers Quick Reference ................................. 693  
F.1 Slurm ................................................................. 693  
F.2 Sun Grid Engine .................................................... 694  
F.3 Torque ................................................................. 696  
F.4 PBS Pro ............................................................... 696  

G Metrics, Health Checks, Enummetrics, And Actions .................. 697  
G.1 Metrics And Their Parameters .................................... 697  
G.1.1 Regular Metrics .................................................. 697  
G.1.2 NFS Metrics ....................................................... 703  
G.1.3 Monitoring System Metrics .................................... 704  
G.1.4 GPU Metrics ....................................................... 706  
G.1.5 Job Metrics ......................................................... 707  
G.1.6 Prometheus Metrics .............................................. 712  
G.1.7 Parameters For Metrics ......................................... 714  
G.2 Health Checks And Their Parameters ............................... 717  
G.2.1 Health Checks ..................................................... 717  
G.2.2 Parameters For Health Checks .................................. 723  
G.3 Enummetrics .......................................................... 724  
G.4 Actions And Their Parameters .................................... 725  
G.4.1 Actions ............................................................. 725  
G.4.2 Parameters For A Monitoring Action ............................ 725  

H Workload Manager Configuration Files Updated By CMDaemon ....... 727  
H.1 Slurm ................................................................. 727  
H.2 Grid Engine (SGE/UGE) ............................................. 727  
H.3 Torque ................................................................. 727  
H.4 PBS Pro ............................................................... 727  
H.5 LSF ..................................................................... 728  

I Changing The LDAP Password ........................................... 729  
I.1 Setting A New Password For The LDAP Server .................... 729  
I.2 Setting The New Password In cmd.conf ............................ 729  
I.3 Checking LDAP Access ............................................... 730  

J Tokens ................................................................. 731  

K Understanding Consolidation ......................................... 739  
K.1 Introduction ......................................................... 739  
K.2 What Is Consolidation? .............................................. 739  
K.3 Raw Data And Consolidation ....................................... 739  
K.4 A Demonstration Of The Output .................................... 740
### Table of Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>L. Node Execution Filters And Execution Multiplexers</td>
<td>743</td>
</tr>
<tr>
<td>L.1 Data Producers: Default Configuration For Running And Sampling</td>
<td>744</td>
</tr>
<tr>
<td>L.1.1 Nodes That Data Producers Are Running On By Default—The nodes Command</td>
<td>744</td>
</tr>
<tr>
<td>L.1.2 Nodes That Data Producers Target By Default—The samplenow Command</td>
<td>744</td>
</tr>
<tr>
<td>L.2 Data Producers: Configuration For Running And Targeting</td>
<td>745</td>
</tr>
<tr>
<td>L.2.1 Custom Metrics From The fm.sh Custom Script</td>
<td>745</td>
</tr>
<tr>
<td>L.3 Replacing A Resource With An Explicit Node Specification</td>
<td>747</td>
</tr>
<tr>
<td>L.4 Excessive Sampling</td>
<td>748</td>
</tr>
<tr>
<td>L.5 Not Just For Nodes</td>
<td>749</td>
</tr>
<tr>
<td>L.6 Lua Node Execution Filters</td>
<td>749</td>
</tr>
</tbody>
</table>
Welcome to the *Administrator Manual* for the Bright Cluster Manager 8.2 cluster environment.

### 0.1 Quickstart

For readers who want to get a cluster up and running as quickly as possible with Bright Cluster Manager, there is a quickstart installation guide in Chapter 1 of the *Installation Manual*.

### 0.2 About This Manual

The rest of this manual is aimed at helping system administrators configure, understand, and manage a cluster running Bright Cluster Manager so as to get the best out of it.

The *Administrator Manual* covers administration topics which are specific to the Bright Cluster Manager environment. Readers should already be familiar with basic Linux system administration, which the manual does not generally cover. Aspects of system administration that require a more advanced understanding of Linux concepts for clusters are explained appropriately.

This manual is not intended for users interested only in interacting with the cluster to run compute jobs. The *User Manual* is intended to get such users up to speed with the user environment and workload management system.

### 0.3 About The Manuals In General

Regularly updated versions of the Bright Cluster Manager 8.2 manuals are available on updated clusters by default at `/cm/shared/docs/cm`. The latest updates are always online at [http://support.brightcomputing.com/manuals](http://support.brightcomputing.com/manuals).

- The *User Manual* describes the user environment and how to submit jobs for the end user.
- The *Cloudbursting Manual* describes how to deploy the cloud capabilities of the cluster.
- The *Developer Manual* has useful information for developers who would like to program with Bright Cluster Manager.
- The *OpenStack Deployment Manual* describes how to deploy OpenStack with Bright Cluster Manager.
- The *Machine Learning Manual* describes how to install and configure machine learning capabilities with Bright Cluster Manager.

If the manuals are downloaded and kept in one local directory, then in most pdf viewers, clicking on a cross-reference in one manual that refers to a section in another manual opens and displays that section in the second manual. Navigating back and forth between documents is usually possible with keystrokes or mouse clicks.

For example: `<Alt>-<Backarrow>` in Acrobat Reader, or clicking on the bottom leftmost navigation button of xpdf, both navigate back to the previous document.

The manuals constantly evolve to keep up with the development of the Bright Cluster Manager environment and the addition of new hardware and/or applications. The manuals also regularly incorporate
customer feedback. Administrator and user input is greatly valued at Bright Computing. So any comments, suggestions or corrections will be very gratefully accepted at manuals@brightcomputing.com.

There is also a feedback form available via Bright View, via the Account icon, following the clickpath:

Account → Help → Feedback

0.4 Getting Administrator-Level Support
If the reseller from whom Bright Cluster Manager was bought offers direct support, then the reseller should be contacted.

Otherwise the primary means of support is via the website https://support.brightcomputing.com. This allows the administrator to submit a support request via a web form, and opens up a trouble ticket. It is a good idea to try to use a clear subject header, since that is used as part of a reference tag as the ticket progresses. Also helpful is a good description of the issue. The followup communication for this ticket typically goes via standard e-mail. Section 13.2 has more details on working with support.

0.5 Getting Professional Services
Bright Computing normally differentiates between professional services (customer asks Bright Computing to do something or asks Bright Computing to provide some service) and support (customer has a specific question or problem that requires an answer or resolution). Professional services can be provided after consulting with the reseller, or the Bright account manager.
Introduction

1.1 Bright Cluster Manager Functions And Aims
Bright Cluster Manager contains tools and applications to facilitate the installation, administration, and monitoring of a cluster. In addition, Bright Cluster Manager aims to provide users with an optimal environment for developing and running applications that require extensive computational resources.

1.2 The Scope Of The Administrator Manual (This Manual)
The Administrator Manual covers installation, configuration, management, and monitoring of Bright Cluster Manager, along with relevant background information to help understand the topics covered.

1.2.1 Installation
Installation can generally be divided into parts as follows, with some parts covered by the Administrator Manual, some by the Installation Manual, and some by other manuals:

• **Initial installation of Bright Cluster Manager**: This is covered in the Installation Manual, which gives a short introduction to the concept of a cluster along with details on installing Bright Cluster Manager onto the head node. The Installation Manual is therefore the first manual an administrator should usually turn to when getting to work with Bright Cluster Manager for the first time. The Administrator Manual can be referred to as the main reference resource once the head node has had Bright Cluster Manager installed on it.

• **Provisioning installation**: This is covered in the Administrator Manual. After the head node has had Bright Cluster Manager installed on it, the other, regular, nodes can (network) boot off it and provision themselves from it with a default image, without requiring a Linux distribution DVD themselves. The network boot and provisioning process for the regular nodes is described in detail in Chapter 5.

  In brief, provisioning installs an operating system and files on a node. This kind of installation to a regular node differs from a normal Linux installation in several ways. An important difference is that content that is put on the filesystem of the regular node is normally overwritten by provisioning when the regular node reboots.

• **Post-installation software installation**: The installation of software to a cluster that is already configured and running Bright Cluster Manager is described in detail in Chapter 11 of this manual.

• **Third-party software installation**: The installation of software that is not developed by Bright Computing, but is supported as a part of Bright Cluster Manager. This is described in detail in the Installation Manual.

• **Cloudbursting, OpenStack, Edge, and Machine Learning**: these are integrated by Bright Computing in various ways. These have their own deployment procedures and have separate manuals.
1.2.2 Configuration, Management, And Monitoring Via Bright Cluster Manager Tools And Applications

The administrator normally deals with the cluster software configuration via a front end to the Bright Cluster Manager. This can be GUI-based (Bright View, section 2.4), or shell-based (cmsh, section 2.5). Other tasks can be handled via special tools provided with Bright Cluster Manager, or the usual Linux tools. The use of Bright Cluster Manager tools is usually recommended over standard Linux tools because cluster administration often has special issues, including that of scale.

The following topics are among those covered in this manual:

<table>
<thead>
<tr>
<th>Chapter, Title</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>2 Cluster Management With Bright Cluster Manager</td>
<td>Introduction to main concepts and tools of Bright Cluster Manager. Lays down groundwork for the remaining chapters</td>
</tr>
<tr>
<td>3 Configuring The Cluster</td>
<td>Further configuration and set up of the cluster after software installation of Bright Cluster Manager on the head node.</td>
</tr>
<tr>
<td>4 Power Management</td>
<td>How power management within the cluster works</td>
</tr>
<tr>
<td>5 Node Provisioning</td>
<td>Node provisioning in detail</td>
</tr>
<tr>
<td>6 User Management</td>
<td>Account management for users and groups</td>
</tr>
<tr>
<td>7 Workload Management</td>
<td>Workload management implementation and use</td>
</tr>
<tr>
<td>8 The cm-scale Service</td>
<td>A Bright Cluster Manager service to dynamically scale the cluster according to need</td>
</tr>
<tr>
<td>9 Containerization</td>
<td>Using Docker and Kubernetes with Bright Cluster Manager</td>
</tr>
<tr>
<td>11 Post-Installation Software Management</td>
<td>Managing, updating, modifying Bright Cluster Manager software and images</td>
</tr>
<tr>
<td>12 Cluster Monitoring</td>
<td>Cluster monitoring, conditional triggering of actions, user portal and job monitoring</td>
</tr>
<tr>
<td>13 Day-To-Day Administration</td>
<td>Miscellaneous administration</td>
</tr>
<tr>
<td>14 High Availability</td>
<td>Background details and setup instructions to build a cluster with redundant head nodes</td>
</tr>
<tr>
<td>15 Dell BIOS Management</td>
<td>BIOS management with Bright Cluster Manager for certain Dell hardware</td>
</tr>
</tbody>
</table>

The appendices to this manual generally give supplementary details to the main text. The following topics are also logically a part of Bright Cluster Manager administration, but they have their own separate manuals. This is because they have, or are eventually expected to have, many features:

- Cloudbursting (Cloudbursting Manual)
- OpenStack deployment (OpenStack Deployment Manual)
- Edge deployment (Edge Manual)
- Developer topics (Developer Manual)

1.3 Outside The Direct Scope Of The Administrator Manual

The following supplementary resources can deal with issues related to this manual, but are outside its direct scope:

- **Use by the end user:** This is covered very peripherally in this manual. The user normally interacts with the cluster by logging into a custom Linux user environment to run jobs. Details on running jobs from the perspective of the user are given in the User Manual.
• **The knowledge base** at http://kb.brightcomputing.com often supplements the *Administrator Manual* with discussion of the following:
  
  – Obscure, or complicated, configuration cases
  – Procedures that are not really within the scope of Bright Cluster Manager itself, but that may come up as part of related general Linux configuration.

• **Further support options.** If the issue is not described adequately in this manual, then section 13.2 describes how to get further support.
Cluster Management With Bright Cluster Manager

This chapter introduces cluster management with Bright Cluster Manager. A cluster running Bright Cluster Manager exports a cluster management interface to the outside world, which can be used by any application designed to communicate with the cluster.

Section 2.1 introduces a number of concepts which are key to cluster management using Bright Cluster Manager.

Section 2.2 gives a short introduction on how the modules environment can be used by administrators. The modules environment provides facilities to control aspects of a users’ interactive sessions and also the environment used by compute jobs.

Section 2.3 introduces how authentication to the cluster management infrastructure works and how it is used. Section 2.4 and section 2.5 introduce the cluster management GUI (Bright View) and cluster management shell (cmsh) respectively. These are the primary applications that interact with the cluster management daemon.

Section 2.6 describes the basics of the cluster management daemon, CMDaemon, running on all nodes of the cluster.

2.1 Concepts

In this section some concepts central to cluster management with Bright Cluster Manager are introduced.

2.1.1 Devices

A device in the Bright Cluster Manager cluster management infrastructure represents components of a cluster. A device can be any of the following types:

- Head Node
- Physical Node
- Virtual Node
- Cloud Node
- GPU Unit
- Chassis
- Ethernet Switch
- InfiniBand Switch

© Bright Computing, Inc.
• Lite Node
• Myrinet Switch
• Power Distribution Unit
• Rack Sensor Kit
• Generic Device

A device can have a number of properties (e.g. rack position, hostname, switch port) which can be set in order to configure the device. Using Bright Cluster Manager, operations (e.g. power on) may be performed on a device. The property changes and operations that can be performed on a device depend on the type of device. For example, it is possible to mount a new filesystem to a node, but not to an Ethernet switch.

Every device that is managed by Bright Cluster Manager has a device state associated with it. The table below describes the most important states for devices:

<table>
<thead>
<tr>
<th>state</th>
<th>device is</th>
<th>monitored by Bright?</th>
<th>state tracking?</th>
</tr>
</thead>
<tbody>
<tr>
<td>UP</td>
<td>UP</td>
<td>monitored</td>
<td>tracked</td>
</tr>
<tr>
<td>DOWN</td>
<td>DOWN</td>
<td>monitored</td>
<td>tracked</td>
</tr>
<tr>
<td>UP/CLOSED</td>
<td>UP</td>
<td>mostly ignored</td>
<td>tracked</td>
</tr>
<tr>
<td>DOWN/CLOSED</td>
<td>DOWN</td>
<td>mostly ignored</td>
<td>tracked</td>
</tr>
</tbody>
</table>

These, and other other states are described in more detail in section 5.5.

DOWN and DOWN/CLOSED states have an important difference. In the case of DOWN, the device is down, but is typically intended to be available, and thus typically indicates a failure. In the case of DOWN/CLOSED, the device is down, but is intended to be unavailable, and thus typically indicates that the administrator would like the device to be ignored.

2.1.2 Software Images

A software image is a blueprint for the contents of the local filesystems on a regular node. In practice, a software image is a directory on the head node containing a full Linux filesystem.

The software image in a standard Bright Cluster Manager installation is based on the same parent distribution that the head node uses. A different distribution can also be chosen after installation, from the distributions listed in section 2.1 of the Installation Manual for the software image. That is, the head node and the regular nodes can run different parent distributions. However, such a “mixed” cluster can be harder to manage and it is easier for problems to arise in such mixtures. Such mixtures, while supported, are therefore not recommended, and should only be administered by system administrators that understand the differences between Linux distributions.

RHEL6/CentOS6/SL6 mixtures are completely compatible with each other on the head and regular nodes. On the other hand, SLES may need some effort to work in a mixture with RHEL/CentOS/SL.

When a regular node boots, the node provisioning system (Chapter 5) sets up the node with a copy of the software image, which by default is called default-image.

Once the node is fully booted, it is possible to instruct the node to re-synchronize its local filesystems with the software image. This procedure can be used to distribute changes to the software image without rebooting nodes (section 5.6.2).

It is also possible to “lock” a software image so that no node is able to pick up the image until the software image is unlocked. (section 5.4.7).

Software images can be changed using regular Linux tools and commands (such as rpm and chroot). More details on making changes to software images and doing image package management can be found in Chapter 11.
2.1.3 Node Categories

The collection of settings in Bright Cluster Manager that can apply to a node is called the configuration of the node. The administrator usually configures nodes using the Bright View (section 2.4) or cmsh (section 2.5) front end tools, and the configurations are managed internally with a database.

A node category is a group of regular nodes that share the same configuration. Node categories allow efficiency, allowing an administrator to:

• configure a large group of nodes at once. For example, to set up a group of nodes with a particular disk layout.

• operate on a large group of nodes at once. For example, to carry out a reboot on an entire category.

A node is in exactly one category at all times, which is default by default. The default category can be changed by accessing the base object of partition mode (page 69), and setting the value of defaultCategory to another, existing, category.

Nodes are typically divided into node categories based on the hardware specifications of a node or based on the task that a node is to perform. Whether or not a number of nodes should be placed in a separate category depends mainly on whether the configuration—for example: monitoring setup, disk layout, role assignment—for these nodes differs from the rest of the nodes.

A node inherits values from the category it is in. Each value is treated as the default property value for a node, and is overruled by specifying the node property value for the node.

One configuration property value of a node category is its software image (section 2.1.2). However, there is no requirement for a one-to-one correspondence between node categories and software images. Therefore multiple node categories may use the same software image, and conversely, one variable image—it is variable because it can be changed by the node setting—may be used in the same node category.

Software images can have their parameters overruled by the category settings. By default, however, the category settings that can overrule the software image parameters are unset.

By default, all nodes are placed in the default category. Alternative categories can be created and used at will, such as:

Example

<table>
<thead>
<tr>
<th>Node Category</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>nodes-ib</td>
<td>nodes with InfiniBand capabilities</td>
</tr>
<tr>
<td>nodes-highmem</td>
<td>nodes with extra memory</td>
</tr>
<tr>
<td>login</td>
<td>login nodes</td>
</tr>
<tr>
<td>storage</td>
<td>storage nodes</td>
</tr>
</tbody>
</table>

2.1.4 Node Groups

A node group consists of nodes that have been grouped together for convenience. The group can consist of any mix of all kinds of nodes, irrespective of whether they are head nodes or regular nodes, and irrespective of what (if any) category they are in. A node may be in 0 or more node groups at one time.

I.e.: a node may belong to many node groups.

Node groups are used mainly for carrying out operations on an entire group of nodes at a time. Since the nodes inside a node group do not necessarily share the same configuration, configuration changes cannot be carried out using node groups.

Example
Node Group | Members
---|---
brokenhardware | node087, node783, node917
headnodes | mycluster-m1, mycluster-m2
rack5 | node212..node254
top | node084, node126, node168, node210

One important use for node groups is in the nodegroups property of the provisioning role configuration (section 5.2.1), where a list of node groups that provisioning nodes provision is specified.

2.1.5 Roles
A role is a task that can be performed by a node. By assigning a certain role to a node, an administrator activates the functionality that the role represents on this node. For example, a node can be turned into provisioning node, or a storage node by assigning the corresponding roles to the node.

Roles can be assigned to individual nodes or to node categories. When a role has been assigned to a node category, it is implicitly assigned to all nodes inside of the category.

A configuration overlay is a feature introduced in Bright Cluster Manager 7.1. A configuration overlay is a group of roles that can be assigned to designated groups of nodes within a cluster.

Some roles allow parameters to be set that influence the behavior of the role. For example, the Slurm Client Role (which turns a node into a Slurm client) uses parameters to control how the node is configured within Slurm in terms of queues and the number of GPUs.

When a role has been assigned to a node category with a certain set of parameters, it is possible to override the parameters for a node inside the category. This can be done by assigning the role again to the individual node with a different set of parameters. Roles that have been assigned to nodes override roles that have been assigned to a node category.

A role can be imported from another entity, such as a role, a category, or a configuration overlay. Examples of role assignment are given in sections 5.2.2 and 5.2.3.

2.2 Modules Environment
The modules environment is a third-party software (section 7.1 of the Installation Manual) that allows users to modify their shell environment using pre-defined modules. A module may, for example, configure the user’s shell to run a certain version of an application.

Details of the modules environment from a user perspective are discussed in section 2.3 of the User Manual. However some aspects of it are relevant for administrators and are therefore discussed here.

2.2.1 Adding And Removing Modules
Modules may be loaded and unloaded, and also be combined for greater flexibility.

Modules currently installed are listed with:

```
module list
```

The modules available for loading are listed with:

```
module avail
```

Loading and removing specific modules is done with `module load` and `module remove`, using this format:

```
module load <MODULENAME1> [<MODULENAME2> ...]
```

For example, loading the shared module (section 2.2.2), the gcc compiler, the openmpi parallel library, and the openblas library, allows an MPI application to be compiled with OpenBLAS optimizations:
Example

module add shared
module add gcc/6.1.0
module add openmpi/gcc/64/1.10.1
module add openblas/dynamic/0.2.18
mpicc -o myapp myapp.c

Specifying version numbers explicitly is typically only necessary when multiple versions of an application are installed and available. When there is no ambiguity, module names without a further path specification may be used.

2.2.2 Using Local And Shared Modules
Applications and their associated modules are divided into local and shared groups. Local applications are installed on the local filesystem, whereas shared applications reside on a shared (i.e. imported) filesystem.

It is recommended that the shared module be loaded by default for ordinary users. Loading it gives access to the modules belonging to shared applications, and allows the module avail command to show these extra modules.

Loading the shared module automatically for root is not recommended on a cluster where shared storage is not on the head node itself. This is because root logins could be obstructed if this storage is not available, and if the root user relies on files in the shared storage.

On clusters without external shared storage, root can safely load the shared module automatically at login. This can be done by running the following command as root:

module initadd shared

Other modules can also be set to load automatically by the user at login by using “module initadd” with the full path specification. With the initadd option, individual users can customize their own default modules environment.

Modules can be combined in meta-modules. By default, the default-environment meta-module exists, which allows the loading of several modules at once by a user. Cluster administrators are encouraged to customize the default-environment meta-module to set up a recommended environment for their users. The default-environment meta-module is empty by default.

The administrator and users have the flexibility of deciding the modules that should be loaded in undecided cases via module dependencies. Dependencies can be defined using the prereq and conflict commands. The man page for modulefile gives details on configuring the loading of modules with these commands.

2.2.3 Setting Up A Default Environment For All Users
How users can set up particular modules to load automatically for their own use with the module initadd command is discussed in section 2.2.2.

How the administrator can set up particular modules to load automatically for all users by default is discussed in this section (section 2.2.3). In this example it is assumed that all users have just the following modules as a default:

Example

[fred@bright82 ~]$ module list
Currently Loaded Modulefiles:
1) shared

The slurm and gdb modules can then be set up by the administrator as a default for all users in the following 2 ways:
1. Creating and defining part of a .profile to be executed for login shells. For example:

   [root@bright82 ~]# cat /etc/profile.d/userdefaultmodules.sh
   module load shared
   module load slurm
   module load gdb

   Whenever users now carry out a bash login, these modules are loaded.

2. Instead of placing the modules directly in a script under profile.d like in the preceding item, a slightly more sophisticated way is to set the modules in the meta-module /cm/shared/modulefiles/default-environment. For example:

   [root@bright82 ~]# cat /cm/shared/modulefiles/default-environment
   #%Module1.0###############################
   ## default modulefile
   ###
   proc ModulesHelp {} {
       puts stderr "$\r\nLoads default environment modules for this cluster"
   }
   module-whatis "adds default environment modules"

   # Add any modules here that should be added by when a user loads the 'default-environment' module
   module add shared slurm gdb

   The script userdefaultmodules.sh script under profile.d then only needs to have the default-environment module loaded in it:

   [root@bright82 ~]# cat /etc/profile.d/userdefaultmodules.sh
   module load default-environment

   Now, whenever the administrator changes the default-environment module, users get these changes too during login.

   The lexicographical order of the scripts in the /etc/profile directory is important. For example, naming the file defaultusermodules.sh instead of userdefaultmodules.sh means that the modules.sh script is run after the file is run, instead of before, which would cause an error.

2.2.4 Creating A Modules Environment Module

All module files are located in the /cm/local/modulefiles and /cm/shared/modulefiles directories. A module file is a Tcl or Lua script in which special commands are used to define functionality. The modulefile(1) man page has more on this.

Cluster administrators can use the existing modules files as a guide to creating and installing their own modules for module environments, and can copy and modify a file for their own software if there is no environment provided for it already by Bright Cluster Manager.

2.2.5 Lua Modules Environment (LMod)

By default, Bright Cluster Manager uses traditional Tcl scripts for its module files, or TMod. Lua modules, or LMod, provide an alternative modules environment, where the files are typically written in Lua. LMod can be used as a replacement for TMod.

Conceptually LMod works in the same way as TMod, but provides some extra features and commands.
For LMod, the module files are typically written in Lua, but LMod is also capable of reading Tcl module files. It is therefore not necessary to convert all existing Tcl modules manually to the Lua language.

On a Bright cluster, both LMod and TMod are installed by default. However only one of them is active, depending on which one is enabled. Switching between LMod and TMod clusterwide can be done by setting an environment variable, $ENABLE_LMOD in the cm-lmod-init.sh shell script:

Example

[root@bright82 ~]# cat /etc/sysconfig/modules/lmod/cm-lmod-init.sh
export ENABLE_LMOD=1

In the preceding example, LMod is enabled, and TMod is disabled if $ENABLE_LMOD is set to 1.

Example

[root@bright82 ~]# cat /etc/sysconfig/modules/lmod/cm-lmod-init.sh
export ENABLE_LMOD=0

In the preceding example, LMod is disabled, and TMod is enabled if $ENABLE_LMOD is set to 0.

A change made in the file is effective after having logged out, then logged into the shell again.

2.3 Authentication

2.3.1 Changing Administrative Passwords On The Cluster

How to set up or change regular user passwords is not discussed here, but in Chapter 6 on user management.

Amongst the administrative passwords associated with the cluster are:

1. The root password of the head node: This allows a root login to the head node.

2. The root passwords of the software images: These allow a root login to a regular node running with that image, and is stored in the image file.

3. The root password of the node-installer: This allows a root login to the node when the node-installer, a stripped-down operating system, is running. The node-installer stage prepares the node for the final operating system when the node is booting up. Section 5.4 discusses the node-installer in more detail.

4. The root password of MySQL: This allows a root login to the MySQL server.

To avoid having to remember the disparate ways in which to change these 4 kinds of passwords, the cm-change-passwd command runs a dialog prompting the administrator on which of them, if any, should be changed, as in the following example:

[root@bright82 ~]# cm-change-passwd

With this utility you can easily change the following passwords:

* root password of head node
* root password of slave images
* root password of node-installer
* root password of mysql

Note: if this cluster has a high-availability setup with 2 head nodes, be sure to run this script on both head nodes.

Change password for root on head node? [y/N]: y
Changing password for root on head node.
Changing password for user root.
New password:
Retype new password:
pwd: all authentication tokens updated successfully.

Change password for root in default-image [y/N]: y
Changing password for root in default-image.
Changing password for user root.
New password:
Retype new password:
pwd: all authentication tokens updated successfully.

Change password for root in node-installer? [y/N]: y
Changing password for root in node-installer.
Changing password for user root.
New password:
Retype new password:
pwd: all authentication tokens updated successfully.

Change password for MYSQL root user? [y/N]: y
Changing password for MYSQL root user.
Old password:
New password:
Re-enter new password:

For a high-availability—also called a failover—configuration, the passwords are copied over automatically to the other head node when a change is made in the software image root password (case 2 on page 11). For the remaining password cases (head root password, MySQL root password, and node-installer root password), the passwords are best “copied” over to the other head node by simply rerunning the script on the other head node.

Also, in the case of the password for software images used by the regular nodes: the new password that is set for a regular node only works on the node after the image on the node itself has been updated, with, for example, the "imageupdate" command (section 5.6.2). Alternatively, the new password can be made to work on the node by simply rebooting the node to pick up the new image.

The LDAP root password is a random string set during installation. Changing this is not done using "cm-change-password". It can be changed as explained in Appendix I.

If the administrator has stored the password to the cluster in the Bright View front-end, then the password should be modified there too (figure 2.1).

2.3.2 Logins Using ssh

The standard system login root password of the head node, the software image, and the node-installer, can be set using the "cm-change-passwd" command (section 2.3.1).

In contrast, ssh logins from the head node to the regular nodes are set by default to be passwordless:

- For non-root users, an ssh passwordless login works if the /home directory that contains the authorized keys for these users is mounted. The /home directory is mounted by default on the head node as well as on the regular node, so that by default a passwordless login works from the head node to the regular nodes, as well as from the regular nodes to the head node.

- For the root user, an ssh passwordless login should always work from the head node to the regular nodes since the authorized keys are stored in /root. Logins from the regular node to the head node are configured by default to request a password, as a security consideration.
2.3 Authentication

Users can be restricted from ssh logins

- on regular nodes using the usernodelogin (section 7.2.1) or User node login (section 7.2.2) settings
- on the head node by modifying the sshd configuration on the head node. For example, to allow only root logins, the value of AllowUsers can be set in /etc/ssh/sshd_config to root. The man page for sshd_config has details on this.

2.3.3 Certificates

PEM Certificates And CMDaemon Front-end Authentication

While nodes in a Bright Cluster Manager cluster accept ordinary ssh based logins, the cluster manager accepts public key authentication using X509v3 certificates. Public key authentication using X509v3 certificates means in practice that the person authenticating to the cluster manager must present their public certificate, and in addition must have access to the private key that corresponds to the certificate.

Bright Cluster Manager uses the PEM format for certificates. In this format, the certificate and private key are stored as plain text in two separate PEM-encoded files, ending in .pem and .key.

Using cmsh and authenticating to the Bright Cluster Manager:  By default, one administrator certificate is created for root for the cmsh front end to interact with the cluster manager. The certificate and corresponding private key are thus found on a newly-installed Bright Cluster Manager cluster on the head node at:

/root/.cm/admin.pem
/root/.cm/admin.key

The cmsh front end, when accessing the certificate and key pair as user root, uses this pair by default, so that prompting for authentication is then not a security requirement. The logic that is followed to access the certificate and key by default is explained in detail in item 2 on page 234.

Using Bright View and authenticating to the Bright Cluster Manager:  When an administrator uses the Bright View front end, a login to the cluster is carried out with username password authentication (figure 2.1), unless the authentication has already been stored in the browser, or unless certificate-based authentication is used.

Certificate-based authentication can be carried out using a PKCS#12 certificate file. This can be generated from the PEM format certificates. For example, for the root user, an openssl command that can be used to generate the admin.pfx file is:

```
openssl pkcs12 -export -in ~/.cm/admin.pem -inkey ~/.cm/admin.key -out ~/.cm/admin.pfx
```

In Chrome, the IMPORT wizard at chrome://settings/certificates can be used to save the file into the browser. For Firefox, the equivalent clickpath is:

```
about:preferences#privacy→Certificates→View Certificates→Your Certificates→Import
```

The browser can then access the Bright View front end without a username/password combination.

If the administrator certificate and key are replaced, then any other certificates signed by the original administrator certificate must be generated again using the replacement, because otherwise they will no longer function.

Certificate generation in general, including the generation and use of non-administrator certificates, is described in greater detail section 6.4.
Replacing A Temporary Or Evaluation License
In the preceding section, if a license is replaced, then regular user certificates need to be generated again. Similarly, if a temporary or evaluation license is replaced, regular user certificates need to be generated again. This is because the old user certificates are signed by a key that is no longer valid. The generation of non-administrator certificates and how they function is described in section 6.4.

2.3.4 Profiles
Certificates that authenticate to CMDaemon contain a profile.

A profile determines which cluster management operations the certificate holder may perform. The administrator certificate is created with the admin profile, which is a built-in profile that allows all cluster management operations to be performed. In this sense it is similar to the root account on unix systems. Other certificates may be created with different profiles giving certificate owners access to a pre-defined subset of the cluster management functionality (section 6.4).

2.4 Bright View GUI
This section introduces the basics of the cluster management GUI (Bright View). Bright View is the web application front end to cluster management in Bright Cluster Manager. It should run without problems on recent (mid-2016 and later) browsers.

2.4.1 Installing The Cluster Management GUI Service
The GUI interface is provided by default as a web service on port 8081 from the head node to the browser. The URL takes the form:

https://<host name or IP address>:8081/bright-view

The Bright Cluster Manager package that provides the service is bright-view, and is installed by default with Bright Cluster Manager. The service can be disabled by removing the package.

Bright Cluster Manager Bright View Login Window
Figure 2.1 shows the login dialog window for Bright View

Figure 2.1: Editing The Cluster Connection Parameters

Bright Cluster Manager Bright View Default Display On Connection
Clicking on the Login button logs the administrator into the Bright View service on the cluster. By default an overview window is displayed, corresponding to the path Cluster→Partition base (figure 2.2).
2.4 Bright View GUI

2.4.2 Navigating The Cluster With Bright View

Aspects of the cluster can be managed by administrators using Bright View (figure 2.2).

The resource tree, displayed on the left side of the window, consists of available cluster usage concepts such as Provisioning, HPC, OpenStack, Cloud, and Containers. It also has a cluster-centric approach to miscellaneous system concepts such as hardware devices Devices, non-hardware resources such as Identity Management, and Networking.

Selecting a resource opens a window that allows parameters related to the resource to be viewed and managed.

As an example, the Cluster resource can be selected. This opens up the so-called Partition base window, which is essentially a representation of the cluster instance.1

The options within the Partition base window are mapped out in figure 2.3 and summarily described next.

---

1The name Partition base is literally a footnote in Bright Cluster Manager history. It derives from the time that Bright clusters were planned to run in separate partitions within the cluster hardware. The primary cluster was then to be the base cluster, running in the base partition. The advent of Bright Cluster Manager cloud computing options in the form of the Cluster-On-Demand option (Chapter 2 of the CloudBursting Manual), and the Cluster Extension option (Chapter 3 of the CloudBursting Manual) means developing cluster partitions is no longer a priority.
Overview

The Overview window has three scrollable overview panes that show current data for the cluster as a whole:

- The top pane, OVERVIEW, shows some system resource information, such as uptime, memory, swap memory.
  
  The CPU Utilization display is typically an item of great interest to a cluster administrator. Its indicator bar is divided into 4 sections. Hovering the mouse cursor over the bar (figure 2.4) shows the usage segregation:

```
<table>
<thead>
<tr>
<th>Memory</th>
<th>35.35%</th>
</tr>
</thead>
<tbody>
<tr>
<td>Swap memory</td>
<td>0.000% user, 0.000% system, 1.000% other, 99.000% idle</td>
</tr>
<tr>
<td>CPU utilization</td>
<td>99.9999%</td>
</tr>
<tr>
<td>Occupation rate</td>
<td>99.9999%</td>
</tr>
</tbody>
</table>
```

Figure 2.4: Hovering The Mouse Cursor Over The CPU Utilization Bar
The segregation is according to the scheme described in the following table:

<table>
<thead>
<tr>
<th>Bright state</th>
<th>top state</th>
<th>CPU time spent by applications in:</th>
</tr>
</thead>
<tbody>
<tr>
<td>user</td>
<td>us</td>
<td>user space</td>
</tr>
<tr>
<td>system</td>
<td>sy</td>
<td>kernel space</td>
</tr>
<tr>
<td>other</td>
<td>sum of:</td>
<td>sum of these other active states:</td>
</tr>
<tr>
<td></td>
<td>wa</td>
<td>i/o wait</td>
</tr>
<tr>
<td></td>
<td>ni</td>
<td>niced user processes</td>
</tr>
<tr>
<td></td>
<td>hi</td>
<td>hardware IRQ servicing/handling</td>
</tr>
<tr>
<td></td>
<td>si</td>
<td>software IRQ servicing/handling</td>
</tr>
<tr>
<td></td>
<td>st</td>
<td>“stolen time”—forced wait for a virtual CPU while hypervisor services another processor</td>
</tr>
<tr>
<td>idle</td>
<td>id</td>
<td>idling</td>
</tr>
</tbody>
</table>

The Bright Cluster Manager CPU Utilization is based on metrics (section G.1) gathered from all nodes and made into a summary. The summary is based on the CPU states defined for the standard top (1) command, and which are described briefly in the table.

- The middle pane, WORKLOAD, shows workloads that the cluster is managing.
- The bottom pane, DISKS, shows disk mount points and their use.

**Settings**
The Settings window has a number of global cluster properties and property groups. These are loosely grouped as follows:

- Cluster name, Administrator e-mail, Time zone
- Node basename, Node digits
- Name servers, Time servers
- Search domains, Relay Host
- Externally visible IP, Provisioning Node Auto Update Timeout
- BMC Settings: Opens up a window to manage BMC settings
- Default category, Default software image, Default burn configuration
- External network, Management network
- Sign installer certificates
- Failover: Opens up a window to manage failover properties.
- Failover groups: Opens up a window to manage failover groups properties.
- Burn configs: Opens up a window to manage burn configuration settings.
License info
The License info window shows information to do with cluster licensing. A slightly obscure property within this window is version, which refers to the version type of the license. The license for Bright Cluster Manager version 7.0 and above is of a type that is compatible with versions all the way up to 8.2. Bright Cluster Manager license versions from before 7.0 are not compatible. In practice it means that an upgrade from before 7.0, to 7.0 or beyond, requires a license upgrade. Bright Computing must be contacted to arrange the license upgrade.

Version info
The Version info window shows version information for important cluster software components, such as the CMDaemon version, and the Bright Cluster Manager version.

Run command
The Run command option allows a specified command to be run on a selected node of the cluster.

Rack View
The Rack View option displays a view of the rack as defined by node allocations made by the administrator to racks and chassis.

2.5 Cluster Management Shell
This section introduces the basics of the cluster management shell, cmsh. This is the command-line interface to cluster management in Bright Cluster Manager. Since cmsh and Bright View give access to the same cluster management functionality, an administrator need not become familiar with both interfaces. Administrators intending to manage a cluster with only Bright View may therefore safely skip this section.

The cmsh front end allows commands to be run with it, and can be used in batch mode. Although cmsh commands often use constructs familiar to programmers, it is designed mainly for managing the cluster efficiently rather than for trying to be a good or complete programming language. For programming cluster management, the use of Python bindings (Chapter 1 of the Developer Manual) is generally recommended instead of using cmsh in batch mode.

Usually cmsh is invoked from an interactive session (e.g. through ssh) on the head node, but it can also be used to manage the cluster from outside.

2.5.1 Invoking cmsh
From the head node, cmsh can be invoked as follows:

Example

[root@mycluster ~]# cmsh
[mycluster]%

By default it connects to the IP address of the local management network interface, using the default Bright Cluster Manager port. If it fails to connect as in the preceding example, but a connection takes place using cmsh localhost, then the management interface is most probably not up, and bringing the management interface up then allows cmsh to connect to CMDaemon.

Running cmsh without arguments starts an interactive cluster management session. To go back to the unix shell, a user enters quit:

[mycluster]% quit
[root@mycluster ~]#
**Batch Mode And Piping In cmsh**

The `-c` flag allows `cmsh` to be used in batch mode. Commands may be separated using semi-colons:

```bash
[root@mycluster ~]# cmsh -c "main showprofile; device status apc01"

admin

apc01 ................ [ UP ]
[root@mycluster ~]#
```

Alternatively, commands can be piped to `cmsh`:

```bash
[root@mycluster ~]# echo device status | cmsh

apc01 ................ [ UP ]
mycluster ............. [ UP ]
node001 ............... [ UP ]
node002 ............... [ UP ]
switch01 ............. [ UP ]
[root@mycluster ~]#
```

**Dotfiles And /etc/cmshrc File For cmsh**

In a similar way to unix shells, `cmsh` sources dotfiles, if they exist, upon start-up in both batch and interactive mode. In the following list of dotfiles, a setting in the file that is in the shorter path will override a setting in the file with the longer path (i.e.: “shortest path overrides”):

- `~/.cm/cmsh/.cmshrc`
- `~/.cm/.cmshrc`
- `~/.cmshrc`

If there is no dotfile for the user, then, if it exists, the file `/etc/cmshrc`, which is accessible to all users, is sourced, and its settings used. Sourcing settings is convenient when defining command aliases. Command aliases can be used to abbreviate longer commands. For example, putting the following in `.cmshrc` would allow `lv` to be used as an alias for `device list virtualnode`:

**Example**

```bash
alias lv device list virtualnode
```

**Built-in Aliases In cmsh**

The following aliases are built-ins, and are not defined in any `.cmshrc` or `cmshrc` files:

```
[bright82]% alias
alias - goto -
alias .. exit
alias / home
alias ? help
alias ds device status
```

**Options Usage For cmsh**

The options usage information for `cmsh` is obtainable with `cmsh -h` (figure 2.5).
### Usage:

- **cmsh [options]** ............... Connect to localhost using default port
- **cmsh [options]** <-certificate|-i certfile> <-key|-k keyfile> <host[:port]>

  **Connect to a cluster using certificate and key in PEM format**

<table>
<thead>
<tr>
<th>Valid options:</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>--help</td>
<td>-h .................</td>
</tr>
<tr>
<td>--noconnect</td>
<td>-u .............</td>
</tr>
<tr>
<td>--controlflag</td>
<td>-z ..........</td>
</tr>
<tr>
<td>--nocolor</td>
<td>-o ..............</td>
</tr>
<tr>
<td>--color &lt;yes/no&gt; ........</td>
<td>Define usage of colors</td>
</tr>
<tr>
<td>--spool &lt;directory&gt; ..........</td>
<td>Alternative /var/spool/cmd</td>
</tr>
<tr>
<td>--tty</td>
<td>t ...................</td>
</tr>
<tr>
<td>--noredirect</td>
<td>-r .............</td>
</tr>
<tr>
<td>--noquitconfirmation</td>
<td>-Q ....</td>
</tr>
<tr>
<td>--command</td>
<td>-c &lt;&quot;c1; c2; ...&quot;&gt; ..</td>
</tr>
<tr>
<td>--file</td>
<td>-f &lt;filename&gt; ..........</td>
</tr>
<tr>
<td>--echo</td>
<td>-x ..................</td>
</tr>
<tr>
<td>--quit</td>
<td>-q ..................</td>
</tr>
<tr>
<td>--disablemultiline</td>
<td>-m ........</td>
</tr>
</tbody>
</table>

**Figure 2.5: Usage information for cmsh**

### Man Page For cmsh

There is also a man page for cmsh(8), which is a bit more extensive than the help text. It does not however cover the modes and interactive behavior.

#### 2.5.2 Levels, Modes, Help, And Commands Syntax In cmsh

The **top-level** of cmsh is the level that cmsh is in when entered without any options.

To avoid overloading a user with commands, cluster management functionality has been grouped and placed in separate cmsh **mode** levels. Mode levels and associated **objects** for a level make up a hierarchy available below the top-level.

There is an object-oriented terminology associated with managing via this hierarchy. To perform cluster management functions, the administrator descends via cmsh into the appropriate mode and object, and carries out actions relevant to the mode or object.

For example, within user mode, an object representing a user instance, fred, might be added or removed. Within the object fred, the administrator can manage its properties. The properties can be data such as a password fred123, or a home directory /home/fred.

Figure 2.6 shows the top-level commands available in cmsh. These commands are displayed when help is typed in at the top-level of cmsh:
All levels inside cmsh provide these top-level commands.
Passing a command as an argument to help gets details for it:

Example
22 Cluster Management With Bright Cluster Manager

[myheadnode]% help run
Name: run - Execute all commands in the given file(s)
Usage: run [OPTIONS] <filename> [<filename2> ...]
Options: -x, --echo
  Echo all commands
  -q, --quit
  Exit immediately after error
[myheadnode]%

In the general case, invoking help at any mode level or within an object, without an argument, provides two lists:

- Firstly, under the title of Top: a list of top-level commands.
- Secondly, under the title of the level it was invoked at: a list of commands that may be used at that level.

For example, entering session mode and then typing in help displays, firstly, output with a title of Top, and secondly, output with a title of session (some output ellipsized):

Example

[myheadnode]% session
[myheadnode->session]% help
================================ Top ================================
...
================================ session ===============================
id ....................... Display current session id
killsession ............. Kill a session
list ..................... Provide overview of active sessions
[myheadnode->session]%

Navigation Through Modes And Objects In cmsh

- To enter a mode, a user enters the mode name at the cmsh prompt. The prompt changes to indicate that cmsh is in the requested mode, and commands for that mode can then be run.

- To use an object within a mode, the use command is used with the object name. In other words, a mode is entered, and an object within that mode is used. When an object is used, the prompt changes to indicate that that object within the mode is now being used, and that commands are applied for that particular object.

- To leave a mode, and go back up a level, the exit command is used. Similarly, if an object is in use, the exit command exits the object. At the top level, exit has the same effect as the quit command, that is, the user leaves cmsh and returns to the unix shell. The string .. is an alias for exit.

- The path command at any mode depth displays a path to the current mode and object, in a form that is convenient for copying and pasting into cmsh.

- The home command, which is aliased to /, takes the user from any mode depth to the top level.

Example

© Bright Computing, Inc.
2.5 Cluster Management Shell

[bright82]% device
[bright82->device]% list

<table>
<thead>
<tr>
<th>Type</th>
<th>Hostname (key)</th>
<th>MAC</th>
<th>Category</th>
</tr>
</thead>
<tbody>
<tr>
<td>EthernetSwitch</td>
<td>switch01</td>
<td>00:00:00:00:00:00</td>
<td></td>
</tr>
<tr>
<td>HeadNode</td>
<td>bright82</td>
<td>00:0C:29:5B:55:46</td>
<td></td>
</tr>
<tr>
<td>PhysicalNode</td>
<td>node001</td>
<td>00:0C:29:7A:41:78</td>
<td>default</td>
</tr>
<tr>
<td>PhysicalNode</td>
<td>node002</td>
<td>00:0C:29:CC:4F:79</td>
<td>default</td>
</tr>
</tbody>
</table>

[bright82->device]% exit
[bright82]% device
[bright82->device]% use node001
[bright82->device[node001]]% path
home;device;use node001;
[bright82->device[node001]]% home
[bright82]% home;device;use node001 #copy-pasted from path output

A command can also be executed in a mode without staying within that mode. This is done by specifying the mode before the command that is to be executed within that node. Most commands also accept arguments after the command. Multiple commands can be executed in one line by separating commands with semi-colons.

A cmsh input line has the following syntax:

```
<mode> <cmd> <arg> ... <arg>; ... ; <mode> <cmd> <arg> ... <arg>
```

where `<mode>` and `<arg>` are optional. 2

Example

[bright82->network]% device status bright82; list
bright82 ................ [ UP ]

<table>
<thead>
<tr>
<th>Name (key)</th>
<th>Type</th>
<th>Netmask</th>
<th>Base address</th>
<th>Domain name</th>
<th>Ipv6</th>
</tr>
</thead>
<tbody>
<tr>
<td>externalnet</td>
<td>External</td>
<td>16</td>
<td>192.168.1.0</td>
<td>brightcomputing.com</td>
<td>no</td>
</tr>
<tr>
<td>globalnet</td>
<td>Global</td>
<td>0</td>
<td>0.0.0.0</td>
<td>cm.cluster</td>
<td></td>
</tr>
<tr>
<td>internalnet</td>
<td>Internal</td>
<td>16</td>
<td>10.141.0.0</td>
<td>eth.cluster</td>
<td></td>
</tr>
</tbody>
</table>

In the preceding example, while in network mode, the status command is executed in device mode on the host name of the head node, making it display the status of the head node. The list command on the same line after the semi-colon still runs in network mode, as expected, and not in device mode, and so displays a list of networks.

Inserting a semi-colon makes a difference, in that mode is entered, so that the list displays a list of nodes (some output truncated here for convenience):

Example

[bright82->network]% device; status bright82; list
bright82 ................ [ UP ]

<table>
<thead>
<tr>
<th>Type</th>
<th>Hostname (key)</th>
<th>MAC</th>
<th>Category</th>
<th>Ip</th>
<th>Network</th>
<th>Status</th>
</tr>
</thead>
<tbody>
<tr>
<td>HeadNode</td>
<td>bright82</td>
<td>FA:16:3E:C8:06:D1</td>
<td></td>
<td>10.141.255.254</td>
<td>internalnet</td>
<td>[ UP ]</td>
</tr>
<tr>
<td>PhysicalNode</td>
<td>node001</td>
<td>FA:16:3E:A2:9C:87</td>
<td>default</td>
<td>10.141.0.1</td>
<td>internalnet</td>
<td>[ UP ]</td>
</tr>
</tbody>
</table>

2 A more precise synopsis is:

```
[<mode>] [<cmd> [<arg> ... ]] [; ... ; [<mode>] [<cmd> [<arg> ... ]]]
```

© Bright Computing, Inc.
2.5.3 Working With Objects

Modes in cmsh work with associated groupings of data called objects. For instance, device mode works with device objects, and network mode works with network objects. The commands used to deal with objects have similar behavior in all modes. Not all of the commands exist in every mode, and not all of the commands function with an explicit object:

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>use</td>
<td>Use the specified object. I.e.: Make the specified object the current object</td>
</tr>
<tr>
<td>add</td>
<td>Create the object and use it</td>
</tr>
<tr>
<td>assign</td>
<td>Assign a new object</td>
</tr>
<tr>
<td>unassign</td>
<td>Unassign an object</td>
</tr>
<tr>
<td>clear</td>
<td>Clear the values of the object</td>
</tr>
<tr>
<td>clone</td>
<td>Clone the object and use it</td>
</tr>
<tr>
<td>remove</td>
<td>Remove the object</td>
</tr>
<tr>
<td>commit</td>
<td>Commit local changes, done to an object, to CMDaemon</td>
</tr>
<tr>
<td>refresh</td>
<td>Undo local changes done to the object</td>
</tr>
<tr>
<td>list</td>
<td>List all objects at current level</td>
</tr>
<tr>
<td>sort</td>
<td>Sort the order of display for the list command</td>
</tr>
<tr>
<td>format</td>
<td>Set formatting preferences for list output</td>
</tr>
<tr>
<td>foreach</td>
<td>Execute a set of commands on several objects</td>
</tr>
<tr>
<td>show</td>
<td>Display all properties of the object</td>
</tr>
<tr>
<td>swap</td>
<td>Swap (exchange) the names of two objects</td>
</tr>
<tr>
<td>get</td>
<td>Display specified property of the object</td>
</tr>
<tr>
<td>set</td>
<td>Set a specified property of the object</td>
</tr>
<tr>
<td>clear</td>
<td>Set default value for a specified property of the object.</td>
</tr>
<tr>
<td>append</td>
<td>Append a value to a property of the object, for a multi-valued property</td>
</tr>
<tr>
<td>removefrom</td>
<td>Remove a value from a specific property of the object, for a multi-valued property</td>
</tr>
<tr>
<td>modified</td>
<td>List objects with uncommitted local changes</td>
</tr>
<tr>
<td>usedby</td>
<td>List objects that depend on the object</td>
</tr>
<tr>
<td>validate</td>
<td>Do a validation check on the properties of the object</td>
</tr>
<tr>
<td>exit</td>
<td>Exit from the current object or mode level</td>
</tr>
</tbody>
</table>

Working with objects with these commands is demonstrated with several examples in this section.

**Working With Objects:** use, exit

**Example**

```
[mycluster->device]% use node001
[mycluster->device[node001]]% status
node001 ............... [ UP ]
[mycluster->device[node001]]% exit
[mycluster->device]%
```

In the preceding example, use node001 issued from within device mode makes node001 the current object. The prompt changes accordingly. The status command, without an argument, then returns status information just for node001, because making an object the current object makes subsequent commands within that mode level apply only to that object. Finally, the exit command exits the current object level.
Working With Objects: add, commit, remove

The commands introduced in this section have many implicit concepts associated with them. So an illustrative session is first presented as an example. What happens in the session is then explained in order to familiarize the reader with the commands and associated concepts.

Example

```
[mycluster->device]% add physicalnode node100 10.141.0.100
[mycluster->device[node100]]% commit
[mycluster->device[node100]]% category add test-category
[mycluster->category[test-category]]% commit
[mycluster->category[test-category]]% remove test-category
[mycluster->category]% commit
Successfully removed 1 Categories
Successfully committed 0 Categories

[mycluster->category]% device remove node100
[mycluster->category]% device
[mycluster->device]% commit
Successfully removed 1 Devices
Successfully committed 0 Devices
```

add: The add command creates an object within its associated mode, and in cmsh the prompt drops into the object level just created. Thus, at the start in the preceding example, within device mode, a new object, named node100, is added. For this particular object properties can also be set, such as the type (physicalnode), and IP address (10.141.0.100). The node object level ([node100*]) is automatically dropped into from device mode when the add command is executed. After execution, the state achieved is that the object has been created with some properties. However, it is still in a temporary, modified state, and not yet persistent.

*Asterisk tags* in the prompt are a useful reminder of a modified state, with each asterisk indicating a tagged object that has an unsaved, modified property. In this case, the unsaved properties are the IP address setting, the node name, and the node type.

The add command—syntax notes:

In most modes the add command takes only one argument, namely the name of the object that is to be created. However, in device mode an extra object-type, in this case physicalnode, is also required as argument, and an optional extra IP argument may also be specified. The response to “help add” while in device mode gives details:

```
[myheadnode->device]% help add
Name:  add - Create a new device of the given type with specified hostname

Usage:
add <type> <hostname> [ip]

Arguments:
  type
    chassis, genericdevice, gpuunit, litenode, cloudnode, physicalnode, virtualnode, headnode, powerdistributionunit, racksensor, ethernetswitch, ibswitch, myrinetswitch
```

commit: The commit command is a further step that actually saves any changes made after executing a command. In this case, in the second line, it saves the node100 object with its properties. The asterisk tag disappears for the prompt if settings for that mode level or below have been saved.
Conveniently, the top level modes, such as the category mode, can be accessed directly from within this level if the mode is stated before the command. So, stating the mode category before running the add command allows the specified category test-category to be added. Again, the test-category object level within category mode is automatically dropped into when the add command is executed.

The -w|--wait option to commit:

The commit command by default does not wait for a state change to complete. This means that the prompt becomes available right away. This means that it is not obvious that the change has taken place, which could be awkward if scripting with cmsh for cloning (discussed shortly) a software image (section 2.1.2). The -w|--wait option to the commit command works around this issue by waiting for the cloning of the software image to be completed before making the prompt available.

remove: The remove command removes a specified object within its associated mode. On successful execution, if the prompt is at the object level, then the prompt moves one level up. The removal is not actually carried out fully yet; it is only a proposed removal. This is indicated by the asterisk tag, which remains visible, until the commit command is executed, and the test-category removal is saved. The remove command can also remove an object in a non-local mode, if the non-local mode is associated with the command. This is illustrated in the example where, from within category mode, the device mode is declared before running the remove command for node100. The proposed removal is configured without being made permanent, but in this case no asterisk tag shows up in the category mode, because the change is in device mode. To drop into device mode, the mode command “device” is executed. An asterisk tag then does appear, to remind the administrator that there is still an uncommitted change (the node that is to be removed) for the mode. The commit command would remove the object whichever mode it is in—the non-existence of the asterisk tag does not change the effectiveness of commit.

The -d|--data option to remove:

The remove command by default removes an object, and not the represented data. An example is if, in softwareimage mode, a software image is removed with the remove (without options) command. As far as the cluster manager is concerned, the image is removed after running commit. However the data in the directory for that software image is not removed. The -d|--data option to the remove command arranges removal of the data in the directory for the specified image, as well as removal of its associated object.

Working With Objects: clone, modified, swap

Continuing on with the node object node100 that was created in the previous example, it can be cloned to node101 as follows:

Example

[mycluster->device]% clone node100 node101
Warning: The Ethernet switch settings were not cloned, and have to be set manually
[mycluster->device*][node101*]% exit
[mycluster->device*]% modified
State Type Name
----------- ---------------
+ Device node101
[mycluster->device*]% commit
[mycluster->device%]
[mycluster->device*]% remove node100
[mycluster->device*]% commit
[mycluster->device%]

© Bright Computing, Inc.
The modified command is used to check what objects have uncommitted changes, and the new object node101 is seen to be modified, is saved with a commit. The device node100 is then removed by using the remove command. A commit executes the removal.

The “+” entry in the State column in the output of the modified command in the preceding example indicates the object is a newly added one, but not yet committed. Similarly, a “-” entry indicates an object that is to be removed on committing, while a blank entry indicates that the object has been modified without an addition or removal involved.

Cloning an object is a convenient method of duplicating a fully configured object. When duplicating a device object, cmsh will attempt to automatically assign a new IP address using a number of heuristics. In the preceding example, node101 is assigned IP address 10.141.0.101.

Sometimes an object may have been misnamed, or physically swapped. For example, node001 exchanged physically with node002 in the rack, or the hardware device eth0 is misnamed by the kernel front end rather than changing the physical device or adjust kernel configurations. This is equivalent to exchanging all the attributes from one name to the other.

For example, if the two interfaces on the head node need to have their names exchanged, it can be done as follows:

```
[mycluster->device]% use mycluster
[mycluster->device[mycluster]]% interfaces
[mycluster->device[mycluster]->interfaces]% list
Type Network device name IP Network
---------- ------------------------ ------------------
physical eth0 [dhcp] 10.150.4.46 externalnet
physical eth1 [prov] 10.141.255.254 internalnet
[bright82->device[mycluster]->interfaces]% swap eth0 eth1; commit
[mycluster->device[mycluster]->interfaces]% list
Type Network device name IP Network
---------- ------------------------ ------------------
physical eth0 [prov] 10.141.255.254 internalnet
physical eth1 [dhcp] 10.150.4.46 externalnet
```

Working With Objects: get, set, refresh

The get command is used to retrieve a specified property from an object, and set is used to set it:

Example

```
[mycluster->device]% use node101
[mycluster->device[node101]]% get category
test-category
[mycluster->device[node101]]% set category default
[mycluster->device*[node101*]]% get category
default
[mycluster->device*[node101*]]% modified
State Type Name
--------- -------- ------------
Device node101
[mycluster->device*[node101*]]% refresh
[mycluster->device[node101]]% modified
No modified objects of type device
[mycluster->device[node101]]% get category
test-category
[mycluster->device[node101]]% exit
```

© Bright Computing, Inc.
Here, the category property of the node101 object is retrieved by using the get command. The property is then changed using the set command. Using get confirms that the value of the property has changed, and the modified command reconfirms that node101 has local uncommitted changes. The refresh command undoes the changes made, and the modified command confirms that no local changes exist. Finally the get command reconfirms that no local changes exist.

A string can be set as a revision label for any object:

Example

\[
\text{[mycluster->device[node101]]\% set revision "changed on 10th May"}
\]
\[
\text{[mycluster->device*[node101*]]\% get revision}
\]
\[
\text{[mycluster->device*[node101*]]\% changed on 10th May 2011}
\]

This can be useful when using shell scripts with an input text to label and track revisions when sending commands to cmsh. How to send commands from the shell to cmsh is introduced in section 2.5.1.

Some properties are Booleans. For these, the values “yes”, “1”, “on” and “true” are equivalent to each other, as are their opposites “no”, “0”, “off” and “false”. These values are case-insensitive.

Working With Objects: clear

Example

\[
\text{[mycluster->device]\% set node101 mac 00:11:22:33:44:55}
\]
\[
\text{[mycluster->device*]\% get node101 mac}
\]
\[
\text{00:11:22:33:44:55}
\]
\[
\text{[mycluster->device*]\% clear node101 mac}
\]
\[
\text{[mycluster->device*]\% get node101 mac}
\]
\[
\text{00:00:00:00:00:00}
\]
\[
\text{[mycluster->device*]}
\]

The get and set commands are used to view and set the MAC address of node101 without running the use command to make node101 the current object. The clear command then unset the value of the property. The result of clear depends on the type of the property it acts on. In the case of string properties, the empty string is assigned, whereas for MAC addresses the special value 00:00:00:00:00:00 is assigned.

Working With Objects: list, format, sort

The list command is used to list all device objects. The -f flag takes a format string as argument. The string specifies what properties are printed for each object, and how many characters are used to display each property in the output line. In following example a list of objects is requested, displaying the hostname, ethernetswitch and ip properties for each object.

Example

\[
\text{[bright82->device]\% list -f hostname:14,ethernetswitch:15,ip}
\]
\[
\text{hostname (key) ethernetswitch ip}
\]
\[
\text{------------------ --------------- -------------------}
\]
\[
apc01 10.142.254.1
\]
\[
bright82 switch01:46 10.142.255.254
\]
\[
node001 switch01:47 10.142.0.1
\]
\[
node002 switch01:45 10.142.0.2
\]
\[
switch01 10.142.253.1
\]
\[
\text{[bright82->device]}
\]

Running the list command with no argument uses the current format string for the mode.

Running the format command without arguments displays the current format string, and also displays all available properties including a description of each property. For example (output truncated):
Example

[bright82->device]% format
Current list printing format:
--------------------------------- 
hostname: [10-14]

Valid fields:
-----------
Revision : Entity revision
Type : The type of the device
activation : Date on which node was defined
additionalhostnames : Additional hostnames that should resolve to the interfaces IP address
banks : Number of banks
...

To change the current format string, a new format string can be passed as an argument to format. The print specification of the format command uses the delimiter “:” to separate the parameter and the value for the width of the parameter column. For example, a width of 10 can be set with:

Example

[bright82->device]% format hostname:10
[bright82->device]% list
hostname (key)
--------------
apc01
bright82
node001
node002
switch01

A range of widths can be set, from a minimum to a maximum, using square brackets. A single minimum width possible is chosen from the range that fits all the characters of the column. If the number of characters in the column exceeds the maximum, then the maximum value is chosen. For example:

Example

[bright82->device]% format hostname: [10-14]
[bright82->device]% list
hostname (key)
--------------
apc01
bright82
node001
node002
switch01

The parameters to be viewed can be chosen from a list of valid fields by running the format command without any options, as shown earlier.

Multiple parameters can be specified as a comma-separated list (with a colon-delimited width specification for each parameter). For example:

Example
Cluster Management With Bright Cluster Manager

[bright82->device]% format hostname:[10-14],ethernetswitch:14,ip:20
[bright82->device]% list
hostname (key) ethernetswitch ip
-------------- -------------- --------------------
apc01 10.142.254.1
bright82 switch01:46 10.142.255.254
node001 switch01:47 10.142.0.1
node002 switch01:45 10.142.0.2
switch01 10.142.253.1

The default parameter settings can be restored with the -r|--reset option:

Example

[bright82->device]% format -r
[bright82->device]% format | head -3
Current list printing format:
----------------------------------------
type:22, hostname:[16-32], mac:18, category:[16-32], ip:15, network:[14-32], status:[16-32]
[bright82->device]%

The sort command sets the alphabetical sort order for the output of the list according precedence of the parameters specified.

Example

[bright82->device]% sort type mac
[bright82->device]% list -f type:15,hostname:15,mac
type hostname (key) mac
--------------- --------------- --------------------
HeadNode bright82 08:0A:27:BA:B9:43
PhysicalNode node002 00:00:00:00:00:00
PhysicalNode log001 52:54:00:DE:E3:6B
[bright82->device]% sort type hostname
[bright82->device]% list -f type:15,hostname:15,mac
type hostname (key) mac
--------------- --------------- --------------------
HeadNode bright82 08:0A:27:BA:B9:43
PhysicalNode log001 52:54:00:DE:E3:6B
PhysicalNode node002 00:00:00:00:00:00
[bright82->device]% sort mac hostname
[bright82->device]% list -f type:15,hostname:15,mac
type hostname (key) mac
--------------- --------------- --------------------
PhysicalNode node002 00:00:00:00:00:00
HeadNode bright82 08:0A:27:BA:B9:43
PhysicalNode log001 52:54:00:DE:E3:6B

The preceding sort commands can alternatively be specified with the -s|--sort option to the list command:

[bright82->device]% list -f type:15,hostname:15,mac --sort type,mac
[bright82->device]% list -f type:15,hostname:15,mac --sort type,hostname
[bright82->device]% list -f type:15,hostname:15,mac --sort mac,hostname

© Bright Computing, Inc.
2.5 Cluster Management Shell

Working With Objects: append, removefrom
When dealing with a property of an object that can take more than one value at a time—a list of values—the append and removefrom commands can be used to respectively append to and remove elements from the list. If more than one element is appended, they should be space-separated. The set command may also be used to assign a new list at once, overwriting the existing list. In the following example values are appended and removed from the powerdistributionunits properties of device node001. The powerdistributionunits properties represent the list of ports on power distribution units that a particular device is connected to. This information is relevant when power operations are performed on a node. Chapter 4 has more information on power settings and operations.

Example

[mycluster->device]% use node001
[mycluster->device[node001]]% get powerdistributionunits
apc01:1
[...device[node001]]% append powerdistributionunits apc01:5
[...device[node001]]% get powerdistributionunits
apc01:1 apc01:5
[...device[node001]]% append powerdistributionunits apc01:6
[...device[node001]]% get powerdistributionunits
apc01:1 apc01:5 apc01:6
[...device[node001]]% removefrom powerdistributionunits apc01:5
[...device[node001]]% get powerdistributionunits
apc01:1 apc01:6
[...device[node001]]% set powerdistributionunits apc01:1 apc01:02
[...device[node001]]% get powerdistributionunits
apc01:1 apc01:2

Working With Objects: usedby
Removing a specific object is only possible if other objects do not have references to it. To help the administrator discover a list of objects that depend on (“use”) the specified object, the usedby command may be used. In the following example, objects depending on device apc01 are requested. The usedby property of powerdistributionunits indicates that device objects node001 and node002 contain references to (“use”) the object apc01. In addition, the apc01 device is itself displayed as being in the up state, indicating a dependency of apc01 on itself. If the device is to be removed, then the 2 references to it first need to be removed, and the device also first has to be brought to the CLOSED state (page 192) by using the close command.

Example

[mycluster->device]% usedby apc01
Device used by the following:
<table>
<thead>
<tr>
<th>Type</th>
<th>Name</th>
<th>Parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>Device</td>
<td>apc01</td>
<td>Device is up</td>
</tr>
<tr>
<td>Device</td>
<td>node001</td>
<td>powerDistributionUnits</td>
</tr>
<tr>
<td>Device</td>
<td>node002</td>
<td>powerDistributionUnits</td>
</tr>
</tbody>
</table>

Working With Objects: validate
Whenever committing changes to an object, the cluster management infrastructure checks the object to be committed for consistency. If one or more consistency requirements are not met, then cmsh reports the violations that must be resolved before the changes are committed. The validate command allows an object to be checked for consistency without committing local changes.

Example

© Bright Computing, Inc.
[mycluster->device] % use node001
[mycluster->device[node001]] % clear category
[mycluster->device*[node001*]] % commit

<table>
<thead>
<tr>
<th>Code</th>
<th>Field</th>
<th>Message</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>category</td>
<td>The category should be set</td>
</tr>
<tr>
<td></td>
<td>set category default</td>
<td></td>
</tr>
<tr>
<td></td>
<td>validate</td>
<td>All good</td>
</tr>
<tr>
<td></td>
<td>commit</td>
<td></td>
</tr>
</tbody>
</table>

Working With Objects: show

The show command is used to show the parameters and values of a specific object. For example for the object node001, the attributes displayed are (some output ellipsized):

```
[mycluster->device[node001]] % show
Parameter                  Value
--------------------------------------- ------------------------------------
Activation                 Thu, 03 Aug 2017 15:57:42 CEST
BMC Settings               <submode>
Block devices cleared on next boot
Category                   default
...                        ...
Data node                  no
Default gateway            10.141.255.254 (network: internalnet)
...                        ...
Software image             default-image
Static routes              <0 in submode>
...                        ...
```

Working With Objects: assign, unassign

The assign and unassign commands are analogous to add and remove. The difference between assign and add from the system administrator point of view is that assign sets an object with settable properties from a choice of existing names, whereas add sets an object with settable properties that include the name that is to be given. This makes assign suited for cases where multiple versions of a specific object choice cannot be used.

For example,

- If a node is to be configured to be run with particular Slurm settings, then the node can be assigned an slurmclient role (section 2.1.5) with the assign command. The node cannot be assigned another slurmclient role with other Slurm settings at the same time. Only the settings within the assigned Slurm client role can be changed.

- If a node is to be configured to run with added interfaces eth3 and eth4, then the node can have both physical interfaces added to it with the add command.

The only place where the assign command is currently used within cmsh is within the roles submode, available under the main category mode or the main device mode. Within roles, assign is used for assigning tasklike roles objects.

Working With Objects: import For Roles

The import command is an advanced command that works within a role. It is used to clone roles between entities.

A node inherits all roles from the category and configuration overlay it is a part of.
Example

[root@bright82 ~]# cmsh
[bright82]# device roles node001
[bright82->device[node001]->roles]# list
Name (key)
-----------------------------------
[category:default] cgroupsupervisor
[category:default] slurmclient

If there is a small change to the default roles to be made, only for node001, in slurmclient, then the role can be imported from the category. Importing the role duplicates the object and assigns the duplicated value to node001.

This differs from simply assigning a slurmclient role to node001, because importing provides the values from the category, whereas assigning provides unset values.

After running import, just as for assign, changes to the role made at node001 level stay at that node level, and changes made to the category-level slurmclient role are not automatically inherited by the node001 slurmclient role.

Example

[bright82->device[node001]->roles]# import<TAB><TAB>
beegfs::admon cephmds cloudgateway elasticsearch
... and other available roles ...
[bright82->device[node001]->roles]# import --category default slurmclient
[bright82->device*[node001*]->roles*]# list
Name (key)
-----------------------------------
[category:default] cgroupsupervisor
slurmclient
[bright82->device*[node001*]->roles*]# set slurmclient GPUs 1
[bright82->device*[node001*]->roles*]# commit

The preceding shows that a list of possible roles is prompted for via tab-completion after having typed import.

The import command in roles mode can duplicate any role between any two entities, as indicated by its help text (help import).

2.5.4 Accessing Cluster Settings

The management infrastructure of Bright Cluster Manager is designed to allow cluster partitioning in the future. A cluster partition can be viewed as a virtual cluster inside a real cluster. The cluster partition behaves as a separate cluster while making use of the resources of the real cluster in which it is contained. Although cluster partitioning is not yet possible in the current version of Bright Cluster Manager, its design implications do decide how some global cluster properties are accessed through cmsh.

In cmsh there is a partition mode which will, in a future version, allow an administrator to create and configure cluster partitions. Currently, there is only one fixed partition, called base. The base partition represents the physical cluster as a whole and cannot be removed. A number of properties global to the cluster exist inside the base partition. These properties are referenced and explained in remaining parts of this manual.

Example

[root@myheadnode ~]# cmsh
[myheadnode]# partition use base
[myheadnode->partition[base]]# show

© Bright Computing, Inc.
### Parameter | Value
---|---
Administrator e-mail | <submode>
BMC Settings | Burn configs <2 in submode>
Cluster name | my-cluster
Default burn configuration | default-destructive
Default category | default
Default software image | default-image
External network | externalnet
Externally visible IP | not defined
Failover | <0 in submode>
Headnode | my-headnode
Management network | internalnet
Name | base
Name servers | 192.168.101.1
Name servers from dhcp | 4.2.2.4
No zero conf | no
Node basename | node
Node digits | 3
Notes | <0 bytes>
Provisioning Node Auto Update Timeout | 300
Relay Host | 
Revision | 
Search domains | example.com
Sign installer certificates | AUTO
Time servers | 0.pool.ntp.org,1.pool.ntp.org,2.pool.ntp.org
Time zone | America/Los_Angeles
UCS Profiles | <0 in submode>

### 2.5.5 Advanced cmsh Features

This section describes some advanced features of cmsh and may be skipped on first reading.

#### Command Line Editing

Command line editing and history features from the readline library are available. [http://tiswww.case.edu/php/chet/readline/rluserman.html](http://tiswww.case.edu/php/chet/readline/rluserman.html) provides a full list of key-bindings.

For users who are reasonably familiar with the bash shell running with readline, probably the most useful and familiar features provided by readline within cmsh are:

- tab-completion of commands and arguments
- being able to select earlier commands from the command history using <ctrl>-r, or using the up- and down-arrow keys

#### History And Timestamps

The history command within cmsh explicitly displays the cmsh command history as a list.

The --timestamps|-t option to the history command displays the command history with timestamps.

#### Example

```
% history | tail -3
162 use node001
163 history
164 history | tail -3
```
Mixing cmsh And Unix Shell Commands

It is often useful for an administrator to be able to execute unix shell commands while carrying out cluster management tasks. The cluster manager shell, cmsh, therefore allows users to execute commands in a subshell if the command is prefixed with a “!” character:

Example

```
[mycluster]% !hostname -f
mycluster.cm.cluster
[mycluster]%
```

Executing the `!` command by itself will start an interactive login sub-shell. By exiting the sub-shell, the user will return to the cmsh prompt.

Besides simply executing commands from within cmsh, the output of operating system shell commands can also be used within cmsh. This is done by using the “backtick syntax” available in most unix shells.

Example

```
[mycluster]% device use 'hostname'
[mycluster->device[mycluster]]% status
mycluster ................ [ UP ]
[mycluster->device[mycluster]]%
```

Output Redirection

Similar to unix shells, cmsh also supports output redirection to the shell through common operators such as >, >> and |.

Example

```
[mycluster]% device list > devices
[mycluster]% device status >> devices
[mycluster]% device list | grep node001
```

<table>
<thead>
<tr>
<th>Type</th>
<th>Hostname (key)</th>
<th>MAC (key)</th>
<th>Category</th>
</tr>
</thead>
<tbody>
<tr>
<td>PhysicalNode</td>
<td>node001</td>
<td>00:E0:81:2E:F7:96</td>
<td>default</td>
</tr>
</tbody>
</table>

The ssh Command

The ssh command is run from within the device mode of cmsh. If an ssh session is launched from within cmsh, then it clears the screen and is connected to the specified node. Exiting from the ssh session returns the user back to the cmsh launch point.

Example

```
[bright82]% device ssh node001
<screen is cleared>
<some MOTD text and login information is displayed>
[root@node001 ~]# exit
Connection to node001 closed.
[bright82]% device use bright82
```

© Bright Computing, Inc.
An alternative to running `ssh` within `cmsh` is to launch it in a subshell anywhere from within `cmsh`, by using `!ssh`.

### The `time` Command

The `time` command within `cmsh` is a simplified version of the standard unix `time` command.

On execution of the `time` command, the second command is executed. After execution of the `time` command is complete, the time the second command took to execute is displayed.

**Example**

```
[bright82->device]% time ds node001
node001 .................. [ UP ]
time: 0.108s
```

### The `watch` Command

The `watch` command within `cmsh` is a simplified version of the standard unix `watch` command.

The `watch` command takes as its argument a second command that is to be executed within `cmsh`. On execution of the `watch` command, the second command is executed every 2 seconds by default, and the output of that second command is displayed.

The repeat interval of the `watch` command can be set with the `--interval` or `--n` option. A running `watch` command can be interrupted with a <Ctrl>-c.

**Example**

```
[bright82->device]% watch newnodes
screen clears
Every 2.0s: newnodes Thu Dec 3 13:01:45 2015
No new nodes currently available.
```

**Example**

```
[bright82->device]% watch -n 3 status -n node001,node002
screen clears
Every 3.0s: status -n node001,node002 Thu Jun 30 17:53:21 2016
node001 .................. [ UP ]
node002 .................. [ UP ]
```

### Looping Over Objects With `foreach`

It is frequently convenient to be able to execute a `cmsh` command on several objects at once. The `foreach` command is available in a number of `cmsh` modes for this purpose. A `foreach` command takes a list of space-separated object names (the keys of the object) and a list of commands that must be enclosed by parentheses, i.e.: “(” and “)”. The `foreach` command will then iterate through the objects, executing the list of commands on the iterated object each iteration.
Basic syntax for the `foreach` command: The basic foreach syntax is:

```bash
foreach <object1> <object2> ...( <command1>; <command2> ... )
```

Example

```
[mycluster->device]% foreach node001 node002 (get hostname; status)
node001 ............. [ UP ]
node002 ............. [ UP ]
[mycluster->device]%
```

With the `foreach` command it is possible to perform set commands on groups of objects simultaneously, or to perform an operation on a group of objects. The `range` command (page 40) provides an alternative to it in many cases.

Advanced options for the `foreach` command: The `foreach` command advanced options can be viewed from the help page:

```
[root@bright82 ~]# cmsh -c "device help foreach"
```

The options can be classed as: grouping options (list, type), adding options, conditional options, and looping options.

- Grouping options:
  
  ```
  -n|--nodes, -g|--group, -c|--category, -r|--rack, -h|--chassis, -e|--overlay, -l|--role, -m|--image, -u|--union, -i|--intersection
  -t|--type virtualnode|cloudnode|node|physicalnode|switch|ibswitch|ethernetswitch|racksensor|chassis|powerdistributionunit
  ```

There are two forms of grouping options shown in the preceding text. The first form uses a list of the objects being grouped, while the second form uses the type of the objects being grouped. These options become available according to the `cmsh` mode used.

In the device mode of `cmsh`, for example, the `foreach` command has many grouping options available. If objects are specified with a grouping option, then the specified objects can be looped over.

For example, with the list form, the `--category (-c)` option takes a node category argument (or several categories), while the `--node (-n)` option takes a node-list argument. Node-lists (specification on page 39) can also use the following, more elaborate, syntax:

```
<nodes>,...,<nodes>,<nodes>...<nodes>
```

Example

```
[demo->device]% foreach -c default (status)
node001 ............. [ DOWN ]
node002 ............. [ DOWN ]
[demo->device]% foreach -g rack8 (status)
...
[demo->device]% foreach -n node001,node008..node016,node032 (status)
...
[demo->device]%
```

© Bright Computing, Inc.
With the type form, using the -t|--type option, the literal value to this option must be one of node, cloudnode, virtualnode, and so on.

If multiple grouping options are used, then the union operation takes place by default.

Both grouping option forms are often used in commands other than foreach for node selection.

- **Adding options:** -o|--clone, -a|--add
  
The --clone (-o) option allows the cloning (section 2.5.3) of objects in a loop. In the following example, from device mode, node001 is used as the base object from which other nodes from node022 up to node024 are cloned:

**Example**

```
[bright82->device] $ foreach --clone node001 -n node022..node024 ()
[bright82->device*] $ list | grep node
Type       Hostname (key)        Ip
---------- --------------  -----------
PhysicalNode node001 10.141.0.1
PhysicalNode node022 10.141.0.22
PhysicalNode node023 10.141.0.23
PhysicalNode node024 10.141.0.24
[bright82->device*] $ commit
```

To avoid possible confusion: the cloned objects are merely objects (placeholder schematics and settings, with some different values for some of the settings, such as IP addresses, decided by heuristics). So it is explicitly not the software disk image of node001 that is duplicated by object cloning to the other nodes by this action at this time.

The --add (-a) option creates the device for a specified device type, if it does not exist. Valid types are shown in the help output, and include physicalnode, headnode, ibswitch.

- **Conditional options:** -s|--status, -q|--quitonunknown
  
The --status (-s) option allows nodes to be filtered by the device status (section 2.1.1).

**Example**

```
[bright82->device] $ foreach -n node001..node004 --status UP (get IP)
10.141.0.1
10.141.0.3
```

Since the --status option is also a grouping option, the union operation applies to it by default too, when more than one grouping option is being run.

The --quitonunknown (-q) option allows the foreach loop to be exited when an unknown command is detected.

- **Looping options:** *, -v|--verbose
  
The wildcard character * with foreach implies all the objects that the list command lists for that mode. It is used without grouping options:

**Example**

```
[myheadnode->device] $ foreach * (get ip; status)
10.141.253.1
switch01 ............ [ DOWN ]
10.141.255.254
```
2.5 Cluster Management Shell

myheadnode ........... [ UP ]
10.141.0.1
node001 ............... [ CLOSED ]
10.141.0.2
node002 ............... [ CLOSED ]

[myheadnode->device]%

Another example that lists all the nodes per category, by running the listnodes command within category mode:

Example

[bright82->category]% foreach * (get name; listnodes)
default

<table>
<thead>
<tr>
<th>Type</th>
<th>Hostname</th>
<th>MAC</th>
<th>Category</th>
<th>Ip</th>
<th>Network</th>
<th>Status</th>
</tr>
</thead>
<tbody>
<tr>
<td>PhysicalNode</td>
<td>node001</td>
<td>FA:16:3E:79:4B:77</td>
<td>default</td>
<td>10.141.0.1</td>
<td>internalnet</td>
<td>[ UP ]</td>
</tr>
<tr>
<td>PhysicalNode</td>
<td>node002</td>
<td>FA:16:3E:41:9E:A8</td>
<td>default</td>
<td>10.141.0.2</td>
<td>internalnet</td>
<td>[ UP ]</td>
</tr>
<tr>
<td>PhysicalNode</td>
<td>node003</td>
<td>FA:16:3E:CO:1F:E1</td>
<td>default</td>
<td>10.141.0.3</td>
<td>internalnet</td>
<td>[ UP ]</td>
</tr>
</tbody>
</table>

The --verbose (-v) option displays the loop headers during a running loop with time stamps, which can help in debugging.

Node List Syntax

Node list specifications, as used in the foreach specification and elsewhere, can be of several types. These types are best explained with node list specification examples:

- **adhoc (with a comma):**
  example: node001,node003,node005,node006

- **sequential (with two dots or square brackets):**
  example: node001..node004
  or, equivalently: node00[1-4]
  which is: node001,node002,node003,node004

- **sequential extended expansion (only for square brackets):**
  example: node[001-002]s[001-005]
  which is:
  node001s001,node001s002,node001s003,node001s004,node001s005,
  node002s001,node002s002,node002s003,node002s004,node002s005

- **rack-based:**
  This is intended to hint which rack a node is located in. Thus:
  - example: r[1-2]n[01-03]
    which is: r1n01,r1n02,r1n03,r2n01,r2n02,r2n03
    This might hint at two racks, r1 and r2, with 3 nodes each.
  - example: rack[1-2]node[0-3]
    which is: rack1node01,rack1node02,rack1node03,rack2node01,
    rack2node02,rack2node03
    Essentially the same as the previous one, but for nodes that were named more verbosely.

- **sequential exclusion (negation):**
  example: node001..node005,-node002..node003
  which is: node001,node004,node005
• **sequential stride (every `<stride>` steps):**
  
  example: node00[1..7:2]
  
  which is: node001,node003,node005,node007

• **mixed list:**
  
  The square brackets and the two dots input specification cannot be used at the same time in one argument. Other than this, specifications can be mixed:

  - example: r1n001..r1n003,r2n003
    
    which is: r1n001,r1n002,r1n003,r2n003
  
  - example: r2n003,r[3-5]n0[01-03]
    
    which is: r2n003,r3n001,r3n002,r3n003,r4n001,r4n002,r4n003,r5n001,r5n002,r5n003
  
  - example: node[001-100],-node[004-100:4]
    
    which is: every node in the 100 nodes, except for every fourth node.

**Setting grouping syntax with the `groupingsyntax` command:** “Grouping syntax” here refers to usage of dots and square brackets. In other words, it is syntax of how a grouping is marked so that it is accepted as a list. The list that is specified in this manner can be for input or output purposes.

The `groupingsyntax` command sets the grouping syntax using the following options:

• **bracket**: the square brackets specification.

• **dot**: the two dots specification.

• **auto**: the default. Setting auto means that:

  - either the dot or the bracket specification are accepted as input,
  
  - the dot specification is used for output.

The chosen `groupingsyntax` option can be made persistent by adding it to the cmshrc dotfiles, or to `/etc/cmshrc` (section 2.5.1).

**Example**

```
[root@bright82 ~]# cat .cm/cmsh/.cmshrc

groupingsyntax auto
```

**The range Command**

The range command provides an interactive option to carry out basic `foreach` commands over a grouping of nodes. When the grouping option has been chosen, the `cmsh` prompt indicates the chosen range.

**Example**

```
[bright82->device]% range -n node0[01-24]
[bright82->device{-n node001..024}]%
```

In the preceding example, commands applied at device level will be applied to the range of 24 node objects.

Continuing the preceding session—if a category can be selected with the `-c` option. If the default category just has three nodes, then output displayed could look like:

**Example**

```
```
Values can be set at device mode level for the selected grouping.

**Example**

```
[bright82->device{-c default}]% get revision
```

```
[bright82->device{-c default}]% set revision test
```

```
[bright82->device{-c default}]% get revision
```

```
test
```

```
test
```

Values can also be set within a submode. However, staying in the submode for a full interaction is not possible. The settings must be done by entering the submode via a semi-colon (new command statement continuation on same line) syntax, as follows:

**Example**

```
[bright82->device{-c default}]% roles; assign pbsproclient; commit
```

The range command can be regarded as a modal way to carry out an implicit `foreach` on the grouping object. Many administrators should find it easier than a `foreach`:

**Example**

```
[bright82->device{-c default}]% get ip
```

```
10.141.0.1
10.141.0.2
10.141.0.3
```

```
[bright82->device{-c default}]% ..
```

```
[bright82->device]% foreach -c default (get ip)
```

```
10.141.0.1
10.141.0.2
10.141.0.3
```

Administrators should be aware that running a `pexec` inside a node range is typically not the intention:

**Example**

```
[bright82->device{-c default}]% range -n node[001-100]
```

```
[bright82->device{-n node[001-100]}]% pexec -n node[001-100] hostname
```

The preceding starts 100 `pexec` commands, each running on each of the 100 nodes.

Further options to the `range` command can be seen with the help text for the command (output truncated):

**Example**
The `range` command allows you to set a range of several devices to execute future commands on.

**Usage:**
```
range [OPTIONS] * (command)
range [OPTIONS] <device> [<device> ...] (command)
```

**Options:**
- `--show`: Show the current range
- `--clear`: Clear the range
- `-v`, `--verbose`: Show header before each element

---

### The bookmark and goto Commands

**Bookmarks:** A `bookmark` in cmsh is a location in the cmsh hierarchy. A bookmark can be:

- set with the `bookmark` command
- reached using the `goto` command

A bookmark is set with arguments to the `bookmark` command within cmsh as follows:

- The user can set the current location as a bookmark:
  - by using no argument. This is the same as setting no name for it
  - by using an arbitrary argument. This is the same as setting an arbitrary name for it
- Apart from any user-defined bookmark names, cmsh automatically sets the special name: `"-"`. This is always the previous location in the cmsh hierarchy that the user has just come from.

All bookmarks that have been set can be listed with the `-l|--list` option.

**Reaching a bookmark:** A bookmark can be reached with the `goto` command. The `goto` command can take the following as arguments: a blank (no argument), any arbitrary bookmark name, or `"-"`. The bookmark corresponding to the chosen argument is then reached.

**Example**
```
[mycluster]% device use node001
[mycluster->device[node001]]% bookmark
[mycluster->device[node001]]% bookmark -l
Name          Bookmark
------------- ------------------------
   home;device;use node001;
   -
[mycluster->device[node001]]% home
[mycluster]% goto
[mycluster->device[node001]]% goto -
[mycluster]% goto
[mycluster->device[node001]]% bookmark dn1
[mycluster->device[node001]]% goto -
[mycluster]% goto dn1
[mycluster->device[node001]]%
```
Saving bookmarks, and making them persistent: Bookmarks can be saved to a file, such as mysaved, with the \texttt{-s|--save} option, as follows:

Example

\texttt{[mycluster]\% bookmark -s mysaved}

Bookmarks can be made persistent by setting (. \texttt{cmshrc} files (page 19) to load a previously-saved bookmarks file whenever a new \texttt{cmsh} session is started. The \texttt{bookmark} command loads a saved bookmark file using the \texttt{-x|--load} option.

Example

\texttt{[root@bright82 ~]\# cat .cm/cmsh/.cmshrc}
\texttt{bookmark -x mysaved}

Renaming Nodes With The \texttt{rename} Command

Nodes can be renamed globally from within partition mode, in the Node basename field associated with the prefix of the node in Bright View (section 3.1.1) or in \texttt{cmsh} (section 2.5.4, and also page 71).

However, a more fine-grained batch renaming is also possible with the \texttt{rename} command, and typically avoids having to resort to scripting mechanisms. Using \texttt{rename} is best illustrated by examples:

The examples begin with using the default basename of \texttt{node} and default node digits (padded suffix number length) of 3.

A simple rename that is a prefix change, can then be carried out as:

Example

\texttt{[bright82->device]\% rename node001..node003 test}
\texttt{Renamed: node001 to test1}
\texttt{Renamed: node002 to test2}
\texttt{Renamed: node003 to test3}

The rename starts up its own numbering from 1, independent of the original numbering. The change is committed using the \texttt{commit} command.

Zero-padding occurs if the number of nodes is sufficiently large to need it. For example, if 10 nodes are renamed (some output elided):

Example

\texttt{[bright82->device]\% rename node[001-010] test}
\texttt{Renamed: node001 to test01}
\texttt{Renamed: node002 to test02}
\texttt{...}
\texttt{Renamed: node009 to test09}
\texttt{Renamed: node010 to test10}

then 2 digits are used for each number suffix, in order to match the size of the last number.

String formatting can be used to specify the number of digits in the padded number field:

Example

\texttt{[bright82->device]\% rename node[001-003] test\%04d}
\texttt{Renamed: node001 to test0001}
\texttt{Renamed: node002 to test0002}
\texttt{Renamed: node003 to test0003}

The target names can conveniently be specified exactly. It requires an exact name mapping. That is, it assumes the source list size and target list size match:

© Bright Computing, Inc.
Example

[bright82->device]% rename node[001-005] test0[1,2,5-7]
Renamed: node001 to test01
Renamed: node002 to test02
Renamed: node003 to test05
Renamed: node004 to test06
Renamed: node005 to test07

The hostnames are sorted alphabetically before they are applied, with some exceptions based on the listing method used.

A --dry-run option can be used to show how the devices will be renamed. Alternatively, the refresh command can clear a proposed set of changes before a commit command commits the change, although the refresh would also remove other pending changes.

Exact name mapping could be used to allocate individual servers to several people:

Example

[root@bright82 ~]# cmsh
[bright82]% device
[bright82->device]% rename node[001-004] alice,bob,charlie,dave
Renamed: node001 to alice
Renamed: node002 to bob
Renamed: node003 to charlie
Renamed: node004 to dave
[bright82->device]% commit

Skipping by a number of nodes is possible using a colon (:). An example might be to skip by two so that twin servers can be segregated into left/right.

Example

[root@bright82 ~]# cmsh
[bright82]% device
[bright82->device]% rename node[001-100:2] left[001-050]
Renamed: node001 to left001
Renamed: node003 to left002
... Renamed: node097 to left049
Renamed: node099 to left050
[bright82->device]% rename node[002-100:2] right[001-050]
Renamed: node002 to right001
Renamed: node004 to right002
... Renamed: node098 to right049
Renamed: node100 to right050
[bright82->device]% commit

2.6 Cluster Management Daemon

The cluster management daemon or CMDaemon is a server process that runs on all nodes of the cluster (including the head node). The cluster management daemons work together to make the cluster manageable. When applications such as cmsh and Bright View communicate with the cluster, they are actually interacting with the cluster management daemon running on the head node. Cluster management applications never communicate directly with cluster management daemons running on non-head nodes.

© Bright Computing, Inc.
The CMDaemon application starts running on any node automatically when it boots, and the application continues running until the node shuts down. Should CMDaemon be stopped manually for whatever reason, its cluster management functionality becomes unavailable, making it hard for administrators to manage the cluster. However, even with the daemon stopped, the cluster remains fully usable for running computational jobs using a workload manager.

The only route of communication with the cluster management daemon is through TCP port 8081. The cluster management daemon accepts only SSL connections, thereby ensuring all communications are encrypted. Authentication is also handled in the SSL layer using client-side X509v3 certificates (section 2.3).

On the head node, the cluster management daemon uses a MySQL database server to store all of its internal data. Monitoring data is also stored in a MySQL database.

2.6.1 Controlling The Cluster Management Daemon

It may be useful to shut down or restart the cluster management daemon. For instance, a restart may be necessary to activate changes when the cluster management daemon configuration file is modified. The cluster management daemon operation can be controlled through the following init script arguments to service cmd. The cmdeamonctl command also works with some arguments:

<table>
<thead>
<tr>
<th>service cmd Arguments</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>stop</td>
<td>stop the cluster management daemon</td>
</tr>
<tr>
<td>start</td>
<td>start the cluster management daemon</td>
</tr>
<tr>
<td>reload</td>
<td>reload configuration of the cluster management daemon</td>
</tr>
<tr>
<td>force-reload</td>
<td>force reload configuration of the cluster management daemon</td>
</tr>
<tr>
<td>restart</td>
<td>restart the cluster management daemon</td>
</tr>
<tr>
<td>try-restart</td>
<td>try to restart the cluster management daemon, but only if it is running</td>
</tr>
<tr>
<td>status</td>
<td>report whether cluster management daemon is running</td>
</tr>
<tr>
<td>full-status*</td>
<td>report detailed statistics about cluster management daemon</td>
</tr>
<tr>
<td>upgrade*</td>
<td>update database schema after version upgrade (expert only)</td>
</tr>
<tr>
<td>debugon*</td>
<td>enable debug logging (expert only)</td>
</tr>
<tr>
<td>debugoff*</td>
<td>disable debug logging (expert only)</td>
</tr>
<tr>
<td>logconf*</td>
<td>Reload log configuration</td>
</tr>
</tbody>
</table>

* arguments that work with cmdeamonctl as well as with service cmd

Example

Restarting the cluster management daemon on the head node of a cluster:

```
[root@mycluster ~]# service cmd restart
Redirecting to /bin/systemctl restart cmd.service
[root@mycluster ~]#
```

Example

Viewing the resources used by CMDaemon, and some other useful information:

```
[root@bright82 etc]# service cmd full-status
  CMDaemon version 1.9 is running (active).
  Running locally

  Startup Time: Sun, 08 Sep 2019 22:37:13 CEST
```
Uptime: 1w 4d

CPU Usage: 3308.37u 1087.5s (0.4%)
Memory Usage: 872MB

Sessions Since Startup: 749
Active Sessions: 5

Number of occupied worker-threads: 1
Number of free worker-threads: 17

Connections handled: 51673
Requests processed: 221655
Total read: 96.1MB
Total written: 267MB

Average request rate: 0.2requests/s
Average bandwidth usage: 267.498B/s

Example

Restarting the cluster management daemon on a sequence of regular nodes, node001 to node040, of a cluster:

[root@mycluster ~]# pdsh -w node00[1-9],node0[1-3][0-9],node040 service cmd restart

This uses pdsh, the parallel shell command (section 13.1).

2.6.2 Configuring The Cluster Management Daemon

Many cluster configuration changes can be done by modifying the cluster management daemon configuration file. For the head node, the file is located at:

/cm/local/apps/cmd/etc/cmd.conf

For regular nodes, it is located inside of the software image that the node uses.

Appendix C describes the supported configuration file directives and how they can be used. Normally there is no need to modify the default settings.

After modifying the configuration file, the cluster management daemon must be restarted to activate the changes.

2.6.3 CMDaemon Versions

Updating CMDaemon

CMDaemon can be updated on the head node with a package manager command such as:

yum update cmdaemon

and on a regular node image with a command such as:

yum update --installroot=/cm/images/<software image> cmdaemon

Updating software on the cluster is covered in greater detail in Chapter11.
CMDaemon Version Extraction
For debugging an issue, knowing the version of CMDaemon that is in use on the cluster can be helpful. The \texttt{cmdaemonversions} command runs within the \texttt{device} mode of \texttt{cmsh}. It lists the CMDaemon version running on the nodes of the cluster.

\textbf{Example}

\begin{verbatim}
[bright82->device]\% cmdaemonversions
Hostname  Version index  Version hash
---------  --------------  ----------------
bright82   146,315       dba359716a
node001    146,315       dba359716a
node002    146,315       dba359716a
\end{verbatim}

A higher version index value indicates a more recent CMDaemon version. The \texttt{--join} option is a formatting option which gathers together versions with the same option:

\begin{verbatim}
[bright82->device]\% cmdaemonversions --join
Version index  Version hash  Count Hostnames
----------------  -------------  -------- ------------------------
146,315          dba359716a   3       node001,node002,bright82
\end{verbatim}

2.6.4 Configuring The Cluster Management Daemon Logging Facilities
CMDaemon generates log messages in \texttt{/var/log/cmdaemon} from specific internal subsystems, such as Workload Management, Service Management, Monitoring, Certs, and so on. By default, none of those subsystems generate detailed (debug-level) messages, as that would make the log file grow rapidly.

CMDaemon Logging Configuration Global Debug Mode
A global debug mode can be enabled in CMDaemon using \texttt{cmdaemonctl}:

\textbf{Example}

\begin{verbatim}
[root@bright82 ~]\# cmdaemonctl -h
cmdaemonctl [OPTIONS...] COMMAND ...

-h --help       Show this help

Commands:
degubon          Turn on CMDaemon debug
deguboff         Turn off CMDaemon debug
...

[root@bright82 ~]\# cmdaemonctl debugon
CMDaemon debug level on

Stopping debug level logs from running for too long by executing \texttt{cmdaemonctl debugoff} is a good idea, especially for production clusters. This is important in order to prevent swamping the cluster with unfeasibly large logs.

CMDaemon Subsystem Logging Configuration Debug Mode
CMDaemon subsystems can generate debug logs separately per subsystem, including by severity level. This can be done by modifying the logging configuration file at:

\texttt{/cm/local/apps/cmd/etc/logging.cmd.conf}
Cluster Management With Bright Cluster Manager

Within this file, a section with a title of #Available Subsystems lists the available subsystems that can be monitored. These subsystems include MON (for monitoring), DB (for database), HA (for high availability), CERTS (for certificates), CEPH (for Ceph), OPENSTACK (for OpenStack), and so on.

CMDaemon Subsystem Logging Configuration Severity Levels

The debug setting is one of several severity levels. Other severity levels are info, warning, error, and all.

Further details on setting subsystem options are given within the logging.cmd.conf file.

For example, to set CMDaemon log output for Monitoring, at a severity level of warning, the file contents for the section severity might look like:

Example

```
Severity {
    warning: MON
}
```

CMDaemon Subsystem Logging Configuration Deployment

The new logging configuration can be reloaded from the file by restarting CMDaemon:

Example

```
[root@bright82 etc]# service cmd restart
```

or by reloading the logging configuration:

Example

```
[root@bright82 etc]# service cmd logconf
```

2.6.5 Configuration File Modification, And The FrozenFile Directive

As part of its tasks, the cluster management daemon modifies a number of system configuration files. Some configuration files are completely replaced, while other configuration files only have some sections modified. Appendix A lists all system configuration files that are modified.

A file that has been generated entirely by the cluster management daemon contains a header:

```
# This file was automatically generated by cmd. Do not edit manually!
```

Such a file will be entirely overwritten, unless the FrozenFile configuration file directive (Appendix C, page 644) is used to keep it frozen.

Sections of files that have been generated by the cluster management daemon will read as follows:

```
# This section of this file was automatically generated by cmd. Do not edit manually!
# BEGIN AUTOGENERATED SECTION -- DO NOT REMOVE
...
# END AUTOGENERATED SECTION -- DO NOT REMOVE
```

Such a file has only the auto-generated sections entirely overwritten, unless the FrozenFile configuration file directive is used to keep these sections frozen.

The FrozenFile configuration file directive in cmd.conf is set as suggested by this example:

Example

```
FrozenFile = { "/etc/dhcpd.conf", "/etc/postfix/main.cf" }
```

If the generated file or section of a file has a manually modified part, and when not using FrozenFile, then during overwriting an event is generated, and the manually modified configuration file is backed up to:

/var/spool/cmd/saved-config-files

Using FrozenFile can be regarded as a configuration technique (section 3.15.3), and one of various possible configuration techniques (section 3.15.1).
2.6 Cluster Management Daemon

2.6.6 Configuration File Conflicts Between The Standard Distribution And Bright Cluster Manager For Generated And Non-Generated Files

While Bright Cluster Manager changes as little as possible of the standard distributions that it manages, there can sometimes be unavoidable issues. In particular, sometimes a standard distribution utility or service generates a configuration file that conflicts with what the configuration file generated by Bright Cluster Manager carries out (Appendix A).

For example, the Red Hat security configuration tool `system-config-securitylevel` can conflict with what `shorewall` (section 7.2 of the Installation Manual) does, while the Red Hat Authentication Configuration Tool `authconfig` (used to modify the `/etc/pam.d/system-auth` file) can conflict with the configuration settings set by Bright Cluster Manager for LDAP and PAM.

In such a case the configuration file generated by Bright Cluster Manager must be given precedence, and the generation of a configuration file from the standard distribution should be avoided. Sometimes using a fully or partially frozen configuration file (section 2.6.5) allows a workaround. Otherwise, the functionality of the Bright Cluster Manager version usually allows the required configuration function to be implemented.

Details on the configuration files installed and updated by the package management system, for files that are “non-generated” (that is, not of the kind in section 2.6.5 or in the lists in Appendixes A.1, A.2 and A.3), are given in Appendix A.4.

2.6.7 CMDaemon Lite

Introduction

As an alternative to the regular CMDaemon, Bright Cluster Manager provides a lightweight CMDaemon, called CMDaemon Lite. This is intended as a minimal alternative to the regular CMDaemon for nodes that are not managed by CMDaemon. CMDaemon Lite is contained in the package `cm-lite-daemon`.

It can be installed on a device where the administrator considers the option of installing a regular, full-featured, CMDaemon to be overkill, but still wants an alternative so that some basic monitoring can be done on the device.

CMDaemon Lite is a Python service, and can be run on a device such as a standalone desktop, running Windows, Linux, or MacOS. It uses up one node license per node that it is run on.

CMDaemon Lite with the standard number of metrics is about 25% lighter on memory resources, and 50% lighter on CPU resources, than the regular CMDaemon.

Deployment

A zipped package can be picked up on the head node from the Bright Cluster Manager repositories with:

Example

```
yum install cm-lite-daemon
```

This places a zip file at `/cm/shared/apps/cm-lite-daemon-dist/cm-lite-daemon.zip`. This file should be moved to and unzipped on the lite node. The lite node is the machine that is to run `cm-lite-daemon`.

```
[root@bright82 ~]# scp /cm/shared/apps/cm-lite-daemon-dist/cm-lite-daemon.zip root@lite01:/opt/cm-lite-daemon.zip
[root@bright82 ~]# ssh root@lite01
[root@lite01 ~]# cd /opt
[root@lite01 opt]# unzip cm-lite-daemon.zip
[root@lite01 opt]# cd cm-lite-daemon
[root@lite01 cm-lite-daemon]# ls -l --group-directories-first
total 44
drwxr-xr-x 7 root root 4096 Nov 11 13:22 cm_lite_daemon
drwxr-xr-x 2 root root 104 Nov 11 13:23 etc
```

© Bright Computing, Inc.
The lite node needs a certificate, and to be registered before \texttt{cm-lite-daemon} can run on it. The easiest way to do this is to use the \texttt{register\_node} utility which is one of the unzipped files in the preceding list. Running it:

- installs required Python packages
- requests a new certificate
- registers the lite node with the head node
- installs \texttt{cm-lite-daemon} as a service.

After \texttt{register\_node} is run, CMDaemon running on the head node is able to see the certificate request. Depending on the network that the CMDaemon Lite on the lite node is connected to, the certificate will be automatically issued, just like it is for regular Bright nodes being installed. However if CMDaemon Lite is connected via different network, then the certificate must be issued manually, which can be done as follows:

Using \texttt{cmsh} the certificate request ID can be found:

\textbf{Example}

\begin{verbatim}
[bright82->cert]% listrequests
Request ID Client type Session ID Name
------------ ------------ ------------ ------------
 1 Lite node .... ....
\end{verbatim}

After finding the correct value for the \texttt{Request ID}, the certificate can then be issued with:

\textbf{Example}

\begin{verbatim}
bright82->cert]% issuecertificate --days 10000
\end{verbatim}

The days field can be used to set how long \texttt{cm-lite-daemon} is allowed to connect. Regular Bright Cluster Manager node certificates have a lifetime of about 10,000 days (about 27 years).

On a Linux machine \texttt{register\_node} starts \texttt{cm-lite-daemon} as a service, so that the following commands work as expected:

\begin{verbatim}
[root@lite01 ~]# service cm-lite-daemon status
...
[root@lite01 ~]# service cm-lite-daemon start
...
[root@lite01 ~]# service cm-lite-daemon stop
...
\end{verbatim}

On non-Linux operating systems, \texttt{cm-lite-daemon} must be started manually.

CMDaemon Lite can be tested by first running it in a foreground shell environment:
2.6 Cluster Management Daemon

Example

[root@lite01 cm-lite-daemon]# ./cm-lite-daemon

The lite node should then show up as being in the UP state in Bright-View or cmsh. Afterwards the cm-lite-daemon Python script can be registered to be autostarted. The administrator should ensure that the running directory for this is set correctly.

The cm-lite-daemon service can alternatively simply be run as a foreground process when needed.

Even Lighter Than CMDaemon Lite: Configuring A Device As A Generic Node

To put things in perspective: so far the options described have been:
1. CMDaemon running on the device
2. CMDaemon Lite running on the device
   A third option that can be considered, is to have
3. no CMDaemon at all running on the device and to register the device as a generic node with the regular CMDaemon on the head node.
   This third option then monitors the device for a basic state of UP/DOWN, but nothing else. In contrast to the first two cases, a node license is not used up.

Even lighter than generic nodes: configuring a device as unmonitored: Devices can alternatively be added to the BIND DNS entries of the zone file via the /var/named/*.include files (Appendix A.1). This is a feature of the Linux operating system rather than a feature of Bright Cluster Manager, and so—perhaps rather obviously—a Bright Cluster Manager node license is also not used up in this case.
   After restarting the named service, the nodes are not seen on the head node, and the device is not monitored in any way. The cluster does however know how to reach it, which in some cases may be all that a cluster administrator wants.
   For example, if a host 10.141.1.20 with hostname myotherhost01, is added to the internalnet network within the domain name eth.cluster, then the session may be run as follows:

Example

[root@head ~]# vi /var/named/eth.cluster.zone.include
(appropriate DNS forward entry is added)
[root@head ~]# cat /var/named/eth.cluster.zone.include
myotherhost01 IN A 10.141.1.20
[root@head ~]# vi /var/named/141.10.in-addr.arpa.zone.include
(appropriate DNS reverse entry is added)
[root@head ~]# cat /var/named/141.10.in-addr.arpa.zone.include
20.1 IN PTR myotherhost01.eth.cluster.
[root@head ~]# service named restart
3

Configuring The Cluster

After Bright Cluster Manager software has been installed on the head node, the cluster must be configured. For convenience, the regular nodes on the cluster use a default software image stored on the head node. The image is supplied to the regular nodes during a process called provisioning (Chapter 5), and both the head node and the regular nodes can have their software modified to suit exact requirements (Chapter 11). This chapter however goes through a number of basic cluster configuration aspects that are required to get all the hardware up and running on the head and regular nodes.

Section 3.1 explains how some of the main cluster configuration settings can be changed.
Section 3.2 details how the internal and external network parameters of the cluster can be changed.
Section 3.3 describes the setting up of network bridge interfaces.
Section 3.4 describes VLAN configuration.
Section 3.5 describes the setting up of network bond interfaces.
Section 3.6 covers how InfiniBand and Omni-Path are set up.
Section 3.7 describes how Baseboard Management Controllers such as IPMI, iLO, DRAC, CIMC, and Redfish are set up.
Section 3.8 describes how switches are set up.
Section 3.9 explains how disk layouts are configured, as well as how diskless nodes are set up.
Section 3.10 describes how NFS volumes are exported from an NFS server and mounted to nodes using the integrated approach of Bright Cluster Manager.
Section 3.11 describes how services can be run from Bright Cluster Manager.
Section 3.12 describes how a rack can be configured and managed with Bright Cluster Manager.
Section 3.13 describes how a GPU unit such as the Dell PowerEdge C410x can be configured with Bright Cluster Manager.
Section 3.14 describes how custom scripts can replace some of the default scripts in use.
Section 3.15 discusses configuration alternatives that are not based on CMDaemon.

More elaborate aspects of cluster configuration such as power management, user management, package management, and workload management are covered in later chapters.
3.1 Main Cluster Configuration Settings

Figure 3.1: Cluster Settings

In Bright View, the Settings window for the Cluster resource (figure 3.1), which represents the overall cluster object, allows changes to be made to some of the main cluster settings. The values can in some cases be overridden by more specific configuration levels, such as category-level or node-level configuration. The main cluster settings in figure 3.1 are related to the cluster name-related settings, cluster networking, and some miscellaneous global cluster settings.

3.1.1 Cluster Configuration: Various Name-Related Settings

In the Settings window, the following settings are used for names throughout the cluster.

- **Cluster name**: (default name: Bright 8.2-stable Cluster).
- **Default category**: (default name: default)
- **Default software image**: (default name: default-image)
- **How the nodes of the cluster are named**:
  - **Node name**: the base prefix, also called *basename* (default prefix name: node)
  - **Node digits size**: number of digits in suffix of node name (default size: 3)
Node names can be managed globally in this window, or in the corresponding partition mode options (page 71) of the cluster object in cmsh. More specific renaming of node names is possible using the rename command from device mode of cmsh, (section 2.5.5, page 43).

3.1.2 Cluster Configuration: Some Network-Related Settings

These following network-related settings are also described in the context of external network settings for the cluster object, in section 3.2.3, as well as in the quickstart in Chapter 1 of the Installation Manual.

Nameserver And Search Domains Used By Cluster

The settings window can be used to set the IP address of the nameserver and the names of the search domains for the cluster.

By default, the nameserver is the internal (regular-nodes-facing) IP address of the head node. Multiple nameservers can be added. If the value is set to 0.0.0.0, then the address supplied via DHCP to the head node is used. Alternatively, a static value can be set.

Instead of using the GUI, the changes to the nameserver and searchdomain values can instead be carried out using cmsh in partition mode (page 69).

The number of names that can be set as search domains used by the cluster has a maximum limit of 6 by default. This is a hardcoded limit imposed by the Linux operating system in older versions.

More recent versions of glibc (glibc > 2.17-222.el7 in RHEL7) no longer set a limit. However, using more than 6 search domains currently requires the use of the GlobalConfig directive, MaximalSearchDomains, in the CMDaemon configuration file (Appendix C). For example, to set 30 domains, the directive setting would be:

```
GlobalConfig = { "MaximalSearchDomains=30" }
```

Changing The Order In resolv.conf

For clusters, CMDaemon by default automatically writes the /etc/resolv.conf by using the following sources, and in the following order:

1. Global network
2. Other networks
3. Category search domains
4. Partition search domains

Because older glibc versions only support 6 entries in /etc/resolv.conf, it is sometimes useful to exclude or reorder the preceding sources.

For a network object, there are two fields that control the position of the domain name in the file /etc/resolv.conf:

**Example**

```
[bright82]% network use ibnet
[bright82->network[ibnet]]% show
...
Exclude from search domain no
Search domain index 0
```

If the `Exclude from search domain` field is set to `yes`, then the domain name for the network is not used.

The `Search domain index` field specifies the position of the domain name. A value of 0 means CMDaemon automatically determines its location.

The index of the category and partition search domains can also be changed by appending a number, `:<index>`, to the domain name:

**Example**
If an index is set for one search domain, then setting indices for all search domains is recommended. Search domains without indices are handled automatically by CMDaemon.

CMDaemon sorts all search domains according to index, and writes `/etc/resolv.conf` with the 6 that have the lowest index, with the lowest index first.

**Externally Visible IP Address**

The externally visible IP address are public (non-RFC 1918) IP addresses to the cluster. These can be set to be visible on the external network.

**Time server(s)**

Time server hostnames can be specified for the NTP client on the head node.

**Time Zone**

The time zone setting for the cluster is applied to the time that the cluster uses.

- In Bright View, the time zone can be selected from a menu displayed by clicking on the **Time zone** menu selection (figure 3.1).

- In `cmsh`, the time zone can be selected in **partition** mode, using the **base** object. Tab-completion prompting after entering "set timezone" displays a list of possible time zones, from which one can be chosen:

  ```
  [bright82]% partition use base
  [bright82->partition[base]]% set timezone america/los_angeles
  [bright82->partition*[base*]]% commit
  ```

**3.1.3 Miscellaneous Settings**

**BMC (IPMI/iLO, DRAC, CIMC, Redfish) Settings**

The BMC (Baseboard Management Controller) access settings can be configured by clicking on the **BMC Settings** option in Bright View. This opens up a subwindow with the following settings:

- **User name:** (default: bright)
- **Password:** (default: random string generated during head node installation)
- **User ID:** (default: 4)
- **Power reset delay:** During a reset, this is the time, in seconds, that the machine is off, before it starts powering up again (default: 0)

BMC configuration is discussed in more detail in section 3.7.

**Administrator E-mail Setting**

By default, the distribution which Bright Cluster Manager runs on sends e-mails for the administrator to the root e-mail address. The administrator e-mail address can be changed within Bright Cluster Manager so that these e-mails are received elsewhere.

- In Bright View, an e-mail address can be set in the **Administrator e-mail** field (figure 3.1).
• In cmsh, the e-mail address (or space-separated addresses) can be set in partition mode, using the base object as follows:

Example

[bright82]% partition use base
[bright82->partition>[base]]% set administratore-mail alf@example.com beth@example.com
[bright82->partition>*[base*]]% commit

The following critical states or errors cause e-mails to be sent to the e-mail address:

• By default, a month before the cluster license expires, a reminder e-mail is sent to the administrator account by CMDaemon. A daily reminder is sent if the expiry is due within a week.
• A service on the head node that fails on its first ever boot.
• When an automatic failover fails on the head or regular node.

SMTP Relay Host Mailserver Setting
The head node uses Postfix as its SMTP server. The default base distribution configuration is a minimal Postfix installation, and so has no value set for the SMTP relay host. To set its value:

• in Bright View: the Relay Host field sets the SMTP relay host for the cluster resource
• in cmsh: the relayhost property can be set for the base object within partition mode:

Example

[root@bright82 ~]# cmsh
[bright82]% partition use base
[bright82->partition>[base]]% set relayhost mail.example.com
[bright82->partition>*[base*]]% commit

Postfix on the regular nodes is configured to use the head node as a relay host and is normally left untouched by the administrator. If the regular node configuration for Postfix is changed in partition mode, then a node reboot deploys the change for the node. Further Postfix changes can be done directly to the configuration files as is done in the standard distribution. The changes must be done after the marked auto-generated sections, and should not conflict with the auto-generated sections.

A convenient way to check mail is functioning is to run Bright Cluster Manager’s testemail command. The command is run from within the main mode of cmsh. It sends a test e-mail out using CM-Daemon:

[root@bright82 ~]# mailq; ls -al /var/spool/mail/root
Mail queue is empty
-rw------- 1 root mail 0 Sep  8 11:11 /var/spool/mail/root
[root@bright82 ~]# cmsh -c "main; testemail"
Mail delivered to postfix
You have new mail in /var/spool/mail/root
[root@bright82 ~]# mailq; ls -al /var/spool/mail/root
Mail queue is empty
-rw------- 1 root mail 749 Sep  8 11:12 /var/spool/mail/root

The test e-mail destination is the administrator e-mail address discussed in the preceding section.
Failover Settings
To access the high availability (HA) feature of the cluster for head nodes, the administrator can click on the Failover option in Bright View. This opens up a subwindow that can be used to configure HA-related parameters (section 14.4.6).

Failover Groups Settings
To access the high availability feature of the cluster for groups of regular nodes, the administrator can click on the Failover groups option in Bright View. This opens up a subwindow that can be used to configure failover-groups-related parameters (section 14.5).

Burn Configs
Burning nodes is covered in Chapter 8 of the Installation Manual. Burn configuration settings for the cluster can be accessed in Bright View via the following clickpaths:

- Cluster[Partition base] → Settings → Default burn configuration
  This allows the Default burn configuration for a node burn run to be modified.

- Cluster[Partition base] → Settings → Burn configs
  This lists the possible burn configuration settings in a subwindow, and allows changes to some of their properties of each item of the list.

3.1.4 Limiting The Maximum Number Of Open Files
Configuring The System Limit On Open Files: The /proc/sys/fs/file-max Setting
The maximum number of open files allowed on a running Linux operating system is determined by /proc/sys/fs/file-max. To configure this setting so that it is persistent, the Linux operating system uses a /etc/sysctl.conf file and *.conf files under /etc/sysctl.d/. Further information on these files can be found via the man page, man(5) sysctl.conf. Bright Cluster Manager adheres to this standard method, and places a settings file 90-cm-sysctl.conf in the directory /etc/sysctl.d.

By default, the value set for file-max by Bright Cluster Manager for versions 8.2 and beyond is 131072. A head node typically is not used to run applications that will exceed this value. However, cluster administrators with larger clusters or with custom needs can change the value according to need.

Configuring The User Limit On Open Files: The nofile Setting
The maximum number of open files allowed for a user can be seen on running ulimit -n. The value is defined by the nofile parameter of ulimit.

By default the value set by Bright Cluster Manager is 131072.

Ulimit limits are limits to restrict the resources used by users. If the pam_limits.so module is used to apply ulimit limits, then the resource limits can be set via the /etc/security/limits.conf file and *.conf files in the /etc/security/limits.d directory. Further information on these files can be found via the man page, man(5) limits.conf.

Resource limits that can be set for user login sessions include the number of simultaneous login sessions, the number of open files, and memory limits.

The maximum number of open files for a user is unlimited by default in an operating system that is not managed by Bright Cluster Manager. However, it is set to 131072 by default for a system managed by Bright Cluster Manager. The nofile value is defined by Bright Cluster Manager in:

- in /etc/security/limits.d/91-cm-limits.conf on the head node
- in /cm/images/<software image name>/etc/security/limits.d/91-cm-limits.conf in the software image that the regular node picks up.

The values set in 91-cm-limits.conf are typically sufficient for a user session, unless the user runs applications that are resource hungry and consume a lot of open files.
Deciding On Appropriate Ulimit, Limit, And System Limit Values

Decreasing the nofile value in /etc/security/limits.d/91-cm-limits.conf (but leaving the /proc/sys/fs/file-max untouched), or increasing /proc/sys/fs/file-max (but leaving the nofile value of 131072 per session as is), may help the system stay under the maximum number of open files allowed.

In general, users should not be allowed to use the head node as a compilation server, or as a testbed, before running their applications. This is because user errors can unintentionally cause the head node to run out of resources and crash it.

Depending what is running on the server, and the load on it, the administrator may wish to increase the resource limit values.

A very rough rule-of-thumb that may be useful as a first approximation to set file-max optimally is suggested in the kernel source code. The suggestion is to simply multiply the system memory (in MB) by 10 per MB, and make the resulting number the file-max value. For example, if the node has 128 GB of memory, then 1280000 can be set as the file-max value.

Fine-tuning to try and ensure that the operating system no longer runs out of file handles, and to try and ensure the memory limits for handling the load are not exceeded, is best achieved via an empirical trial-and-error approach.

3.2 Network Settings

A simplified quickstart guide to setting up the external head node network configuration on a vendor-prepared cluster is given in Chapter 6 of the Installation Manual. This section (3.2) covers network configuration more thoroughly.

After the cluster is set up with the correct license as explained in Chapter 4 of the Installation Manual, the next configuration step is to define the networks that are present (sections 3.2.1 and 3.2.2).

During Bright Cluster Manager installation at least three default network objects were created:

- **internalnet**: the primary internal cluster network, and the default management network. This is used for booting non-head nodes and for all cluster management communication. In the absence of other internal networks, internalnet is also used for storage and communication between compute jobs. Changing the configuration of this network is described on page 70 under the subheading “Changing Internal Network Parameters For The Cluster”.

- **externalnet**: the network connecting the cluster to the outside world (typically a corporate or campus network). Changing the configuration of this network is described on page 66 under the subheading “Changing External Network Parameters For The Cluster”. This is the only network for which Bright Cluster Manager also supports IPv6.

- **globalnet**: the special network used to set the domain name for nodes so that they can be resolved whether they are cloud nodes or not. This network is described further on page 73 under the subheading “Changing The Global Network Parameters For The Cluster”.

For a Type 1 cluster (section 3.3.7 of the Installation Manual) the internal and external networks are illustrated conceptually by figure 3.2.
The configuration of network settings is completed when, after having configured the general network settings, specific IP addresses are then also assigned to the interfaces of devices connected to the networks.

- Changing the configuration of the head node external interface is described on page 67 under the subheading “The IP address of the cluster”.

- Changing the configuration of the internal network interfaces is described on page 71 under the subheading “The IP addresses and other interface values of the internal network”.
  
  - How to set a persistent identity for an interface—for example, to ensure that a particular interface that has been assigned the name eth3 by the kernel keeps this name across reboots—is covered in section 5.8.1, page 206.

- Changing the configuration of globalnet is described on page 73 under the subheading “Changing The Global Network Parameters For The Cluster”. IP addresses are not configured at the globalnet network level itself.

### 3.2.1 Configuring Networks

The network mode in cmsh gives access to all network-related operations using the standard object commands. Section 2.5.3 introduces cmsh modes and working with objects.

In Bright View, a network can be configured via the clickpath Networking→Networks, which opens up the Networks subwindow (figure 3.3):
3.2 Network Settings

In the context of the OSI Reference Model, each network object represents a layer 3 (i.e. Network Layer) IP network, and several layer 3 networks can be layered on a single layer 2 network (e.g. routes on an Ethernet segment).

Selecting a network such as internalnet or externalnet in the resource tree displays, for example, by double-clicking the row, opens up the Overview window of that network. The Overview window by default lists the device names and device properties that are in the network. For example the internal network typically has some nodes, switches, and other devices attached to it, each with their IP addresses and interface (figure 3.4).

Selecting the Settings option opens a scrollable pane that allows a number of network properties to be changed (figure 3.5).
Figure 3.5: Network Settings

The properties of figure 3.5 are introduced in table 3.2.1.
3.2 Network Settings

<table>
<thead>
<tr>
<th>Property</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>name</td>
<td>Name of this network.</td>
</tr>
<tr>
<td>Domain Name</td>
<td>DNS domain associated with the network.</td>
</tr>
<tr>
<td>Type</td>
<td>Menu options to set the network type. Options are Internal, External, Tunnel, Global, Cloud, or NetMap network.</td>
</tr>
<tr>
<td>MTU</td>
<td>Maximum Transmission Unit. The maximum size of an IP packet transmitted without fragmenting.</td>
</tr>
<tr>
<td>Node booting</td>
<td>Enabling means nodes are set to boot from this network (useful in the case of nodes on multiple networks). For an internal subnet called &lt;subnet&gt;, when node booting is set, CMDaemon adds a subnet configuration /etc/dhcpd.&lt;subnet&gt;.conf on the head node, which is accessed from /etc/dhcpd.conf.</td>
</tr>
<tr>
<td>Lock down dhcpd</td>
<td>Enabling means new nodes are not offered a PXE DHCP IP address from this network, i.e. DHCPD is “locked down”. A DHCP “deny unknown-clients” option is set by this action, so no new DHCP leases are granted to unknown clients for the network. Unknown clients are nodes for which Bright Cluster Manager has no MAC addresses associated with the node.</td>
</tr>
<tr>
<td></td>
<td>• It can be set in Bright View via the Networking resource, selecting a network item, and then choosing Node booting from within Settings.</td>
</tr>
<tr>
<td></td>
<td>• It can be set in cmsh via the network mode, selecting a network, and then setting nodebooting to yes.</td>
</tr>
<tr>
<td>Management allowed</td>
<td>Enabling means that the network has nodes managed by the head node.</td>
</tr>
<tr>
<td>Base address</td>
<td>Base address of the network (also known as the network address).</td>
</tr>
<tr>
<td>Broadcast address</td>
<td>Broadcast address of the network.</td>
</tr>
<tr>
<td>Dynamic range start/end</td>
<td>Start/end IP addresses of the DHCP range temporarily used by nodes during PXE boot on the internal network. These are addresses that do not conflict with the addresses assigned and used by nodes during normal use.</td>
</tr>
<tr>
<td>Netmask bits</td>
<td>Prefix-length, or number of bits in netmask. The part after the “/” in CIDR notation.</td>
</tr>
<tr>
<td>Gateway</td>
<td>Default route IP address</td>
</tr>
</tbody>
</table>

Table 3.2.1: Network Configuration Settings

In basic networking concepts, a network is a range of IP addresses. The first address in the range is the base address. The length of the range, i.e. the subnet, is determined by the netmask, which uses CIDR notation. CIDR notation is the so-called / (“slash”) representation, in which, for example, a CIDR notation of 192.168.0.1/28 implies an IP address of 192.168.0.1 with a traditional netmask of 255.255.255.240 applied to the 192.168.0.0 network. The netmask 255.255.255.240 implies that bits 28–32 of the 32-bit dotted-quad number 255.255.255.255 are unmasked, thereby implying a 4-bit-sized host range of 16 (i.e. $2^4$) addresses.

The sipcalc utility installed on the head node is a useful tool for calculating or checking such IP subnet values (man sipcalc or sipcalc -h for help on this utility):

**Example**

```
user@bright82:~ $ sipcalc 192.168.0.1/28
- [ipv4 : 192.168.0.1/28] - 0
```

© Bright Computing, Inc.
Every network has an associated DNS domain which can be used to access a device through a particular network. For internalnet, the default DNS domain is set to eth.cluster, which means that the hostname node001.eth.cluster can be used to access device node001 through the primary internal network. If a dedicated storage network has been added with DNS domain storage.cluster, then node001.storage.cluster can be used to reach node001 through the storage network. Internal DNS zones are generated automatically based on the network definitions and the defined nodes on these networks. For networks marked as external, no DNS zones are generated.

3.2.2 Adding Networks

Once a network has been added, it can be used in the configuration of network interfaces for devices.

In Bright View the Add button in the Networks subwindow (figure 3.3) can be used to add a new network. After the new network has been added, the Settings pane (figure 3.5) can be used to further configure the newly-added network.

In cmsh, a new network can be added from within network mode using the add or clone commands.

The default assignment of networks (internalnet to Management network and externalnet to External network) can be changed using Bright View, via the Settings window of the cluster object (figure 3.1).

In cmsh the assignment to Management network and External network can be set or modified from the base object in partition mode:

**Example**

```
[root@bright82 ~]# cmsh
[bright82% partition use base
[bright82-%partition[base]]% set managementnetwork internalnet; commit
[bright82-%partition[base]]% set externalnetwork externalnet; commit
```
• In Bright View, the click path for modifying the host name is Devices→Head Nodes→Edit→Settings→Hostname→Save. That is, under the Devices resource, the Head Nodes option is selected. The Edit button can then be used to edit the host. This opens up a pane, and the Settings option can then be selected. The Hostname record can then be modified (figure 3.6), and saved by clicking on the Save button. When setting a hostname, a domain is not included.

After the change, as suggested by Bright View, the head node must be rebooted.

![Figure 3.6: Head Node Settings](image)

• In cmsh, the hostname of the head node can also be changed, via device mode:

**Example**

```
[root@bright82 ~]# cmsh
[bright82]# device use bright82
[bright82->device[bright82]]% set hostname foobar
[foobar->device*[foobar*]]% commit
[foobar->device*[foobar*]]
Tue Jan 22 17:35:29 2013 [warning] foobar: Reboot required: Hostname changed
[foobar->device*[foobar*]]% quit
[root@bright82 ~]# sleep 30; hostname -f foobar.cm.cluster
[root@bright82 ~]#
```

The prompt string shows the new hostname after a short time, when a new shell is started.

After the hostname has been set, as suggested by cmsh, the head node must be rebooted.

**Adding Hostnames To The Internal Network**

Additional hostnames, whether they belong to the cluster nodes or not, can be added as name/value pairs to the /etc/hosts file(s) within the cluster. This should be done only outside the specially-marked CMDaemon-managed section. It can be done to the file on the head node, or to the file on the software image for the regular nodes, or to both, depending on where the resolution is required.

However, for hosts that are on the internal network of the cluster, such as regular nodes, it is easier and wiser to avoid adding additional hostnames via /etc/hosts.
Instead, it is recommended to let Bright Cluster Manager manage host name resolution for devices on the internalnet through its DNS server on the internalnet interface. The host names can be added to the additionalhostnames object, from within interfaces submode for the head node. The interfaces submode is accessible from the device mode. Thus, for the head node, with eth1 as the interface for internalnet:

Example

```
[bright82]% device use bright82
[bright82->device[bright82]]% interfaces
[bright82->device[bright82]->interfaces]% use eth1
[bright82->device[bright82]->interfaces[eth1]]% set additionalhostnames test
[bright82->device[bright82]->interfaces[eth1]]% commit
```

Fri Oct 12 16:48:47 2012 [notice] bright82: Service named was restarted

```
[bright82->device[bright82]->interfaces[eth1]]% !ping test
PING test.cm.cluster (10.141.255.254) 56(84) bytes of data.
```

Multiple hostnames can be added as space-separated entries.

The named service automatically restarts within about 20 seconds after committal, implementing the configuration changes. This kind of restart is a feature (section 3.11.1) of changes made to service configurations by Bright View or cmsh.

### Changing External Network Parameters For The Cluster

#### The external network parameters of the cluster:

When a cluster interacts with an external network, such as a company or a university network, its connection behavior is determined by the settings of two objects: firstly, the external network settings of the Networks resource, and secondly, by the cluster network settings.

1. **The external network object** contains the network settings for all objects configured to connect to the external network, for example, a head node. Network settings are configured in Bright View under the Networking resource, then under the Networks subwindow, then within the Settings option for a selected network. Figure 3.5 shows a settings window for when the internalnet item has been selected, but in the current case the externalnet item must be selected instead. The following parameters can then be configured:

   - the IP network parameters of the cluster (but not the IP address of the cluster):
     - **Base address**: the network address of the external network (the “IP address of the external network”). This is not to be confused with the IP address of the cluster, which is described shortly after this.
     - **Broadcast address**: the broadcast address of the external network. This is not to be confused with the IP address of the internal network (page 63).
     - **Dynamic range start** and **Dynamic range end**: Not used by the external network configuration.
     - **Netmask bits**: the netmask size, or prefix-length, of the external network, in bits.
     - **Gateway**: the default route for the external network.
   - the **network name** (what the external network itself is called), by default this is defined as externalnet in the base partition on a newly installed Type 1 cluster,
   - the **Domain Name**: the network domain (LAN domain, i.e. what domain machines on the external network use as their domain),
   - the **External network checkbox**: this is checked for a Type 1 cluster,
• and MTU size (the maximum value for a TCP/IP packet before it fragments on the external network—the default value is 1500).

If the cluster is running the Bright Cluster Manager for OpenStack edition then some OpenStack parameters are also set:

• OpenStack network type: the OpenStack network type. Choices are: None, Flat, VLAN Host, VLAN, Other, VXLAN Host, FloatingIps
• OpenStack DNS Format: the format for instances that have a floating IP address assigned to them.
• OpenStack network is shared: If enabled, the OpenStack network is a shared, snoopable network.
• OpenStack alloc pool start/end: the start/end addresses of the OpenStack allocation.
• OpenStack VLAN Range/ID: The VLAN range and ID (if using VLANs).
• OpenStack Host network: Defaults to vlanhostnet (for VLANs) or vxlanhostnet (for VXLANs)
• OpenStack Physical network name: Name of the OpenStack physical network, defaults to internalnet

2. The cluster object contains other network settings used to connect to the outside. These are configured in the Settings options of the cluster object resource in Bright View (figure 3.1):

• e-mail address(es) for the cluster administrator,
• any additional external name servers used by the cluster to resolve external host names. As an aside: by default, only the head node name server is configured, and by default it only serves hosts on the internal network via the internal interface. Enabling the PublicDNS directive (Appendix C) allows the head node name server to resolve queries about internal hosts from external hosts via any interface, including the external interface.
• the DNS search domain (what the cluster uses as its domain),
• and NTP time servers (used to synchronize the time on the cluster with standard time) and time zone settings.

These settings can also be adjusted in cmsh in the base object under partition mode.

Changing the networking parameters of a cluster (apart from the IP address of the cluster) therefore requires making changes in the settings of the two preceding objects.

The IP address of the cluster: The cluster object itself does not contain an IP address value. This is because it is the cluster network topology type that determines whether a direct interface exists from the cluster to the outside world. Thus, the IP address of the cluster in the Type 1, Type 2, and Type 3 configurations (section 3.3.7 of the Installation Manual) is defined by the cluster interface that faces the outside world. For Type 1, this is the interface of the head node to the external network (figure 3.2). For Type 2 and Type 3 interfaces the cluster IP address is effectively that of an upstream router, and thus not a part of Bright Cluster Manager configuration. Thus, logically, the IP address of the cluster is not a part of the cluster object or external network object configuration.

For a Type 1 cluster, the head node IP address can be set in Bright Cluster Manager, separately from the cluster object settings. This is then the IP address of the cluster according to the outside world.

Setting the network parameters of the cluster and the head node IP address: These values can be set using Bright View or cmsh:
With Bright View: The cluster network object and associated settings are accessed as follows:

The external network object:
The external network object is accessed via the click path Networking → Networks → externalnet → Edit → Settings which reaches the window shown in figure 3.7:

![Network Settings For External Network](image)

The cluster object:
The cluster object and associated settings are accessed as shown in figure 3.1

The head node IP address:
For a head node bright82, where the IP address is used by the interface eth0, the static IP address can be set via the clickpath Devices → Head Nodes → Edit[bright82] → Settings → Interfaces[eth0] → IP (figure 3.8).

With cmsh: The preceding Bright View configuration can also be done in cmsh, using the network, partition and device modes, as in the following example:

Example

[bright82]% network use externalnet
[bright82->network[externalnet]]% set baseaddress 192.168.1.0
3.2 Network Settings

Figure 3.8: Setting The IP Address On A Head Node In Bright View

```bash
[bright82->network*[externalnet*]]% set netmaskbits 24
[bright82->network*[externalnet*]]% set gateway 192.168.1.1
[bright82->network*[externalnet*]]% commit
[bright82->network[externalnet]]% partition use base
[bright82->partition[base]]% set nameservers 192.168.1.1
[bright82->partition*[base*]]% set searchdomains x.com y.com
[bright82->partition*[base*]]% append timeservers ntp.x.com
[bright82->partition*[base*]]% commit
[bright82->partition[base]]% device use bright82
[bright82->device[bright82]]% interfaces
[bright82->device[bright82]->interfaces]% use eth1
[bright82->device[bright82]->interfaces[eth1]]% set ip 192.168.1.176
[bright82->device[bright82]->interfaces*[eth1*]]% commit
[bright82->device[bright82]->interfaces[eth1]]%
```

After changing the external network configurations, a reboot of the head node is necessary to activate the changes.

Using DHCP to supply network values for the external interface: Connecting the cluster via DHCP on the external network is not generally recommended for production clusters. This is because DHCP-related issues can complicate networking troubleshooting compared with using static assignments.

For a Type 1 network, the cluster and head node can be made to use some of the DHCP-supplied external network values as follows:

- In Bright View, the DHCP setting of figure 3.8 can be set to Enabled
- Alternatively, in cmsh, within interfaces mode for the head node interface, the value of the parameter DHCP can be set:

```bash
[bright82->device[bright82]->interfaces[eth0]]% set dhcp yes
```
The gateway address, the name server(s), and the external IP address of the head node are then obtained via a DHCP lease. Time server configuration for externalnet is not picked up from the DHCP server, having been set during installation (figure 3.31 in Chapter 3 of the Installation Manual). The time servers can be changed using Bright View as in figure 3.1, or using cmsh in partition mode as in the preceding example. The time zone can be changed similarly.

It is usually sensible to reboot after implementing these changes in order to test the changes are working as expected.

Changing Internal Network Parameters For The Cluster

When a cluster interacts with the internal network that the regular nodes and other devices are on, its connection behavior with the devices on that network is determined by settings in:

1. the internal network of the Networks resource (page 70)
2. the cluster network for the internal network (page 71)
3. the individual device network interface (page 71)
4. the node categories network-related items for the device (page 72), in the case of the device being a regular node.

In more detail:

1. The internal network object: has the network settings for all devices connecting to the internal network, for example, a login node, a head node via its internalnet interface, or a managed switch on the internal network. In Bright View, the settings can be configured under the Networking resource, then under the Networks subwindow, then within the Settings option for the internalnet item (figure 3.5). In cmsh the settings can be configured by changing the values in the internalnet object within cmsh’s network mode. Parameters that can be changed include:

   - the IP network parameters of the internal network (but not the internal IP address):
     - “Base address”: the internal network address of the cluster (the “IP address of the internal network”). This is not to be confused with the IP address of the internal network interface of the head node. The default value is 10.141.0.0.
     - “Netmask bits”: the netmask size, or prefix-length, of the internal network, in bits. The default value is 16.
     - Gateway: the default gateway route for the internal network. If unset, or 0.0.0.0 (the default), then its value is decided by the DHCP server on the head node, and nodes are assigned a default gateway route of 10.141.255.254, which corresponds to using the head node as a default gateway route for the interface of the regular node. The effect of this parameter is overridden by any default gateway route value set by the value of Default gateway in the node category.
     - “Dynamic range start” and “Dynamic range end”: These are the DHCP ranges for nodes. DHCP is unset by default. When set, unidentified nodes can use the dynamic IP address values assigned to the node by the node-installer. These values range by default from 10.141.128.0 to 10.141.143.255. Using this feature is not generally recommended, in order to avoid the possibility of conflict with the static IP addresses of identified nodes.
     - Node booting: This allows nodes to boot from the provisioning system controlled by CMDaemon. The parameter is normally set for the management network (that is the network over which CMDaemon communicates to manage nodes) but booting can instead be carried out over a separate physical non-management network. Booting over InfiniBand or OmniPath is possible (section 5.1.3).
3.2 Network Settings

- Lock down dhcpd, if set to yes, stops new nodes from booting via the network. New nodes are those nodes which are detected but the cluster cannot identify based on CMDaemon records. Details are given in Chapter 5 about booting, provisioning, and how a node is detected as new.

- the “domain name” of the network. This is the LAN domain, which is the domain machines on this network use as their domain. By default, set to a name based on the network interface type used on the internal network, for example eth.cluster. In any case, the FQDN must be unique for every node in the cluster.

- the network name, or what the internal network is called. By default, set to internalnet.

- The MTU size, or the maximum value for a TCP/IP packet before it fragments on this network. By default, set to 1500.

2. The cluster object: has other network settings that the internal network in the cluster uses. These particulars are configured in the Settings option of the cluster object resource in Bright View (figure 3.1). The cmsh equivalents can be configured from the base object in partition mode. Values that can be set include:

- the “Management network”. This is the network over which CMDaemon manages the nodes. Management means that CMDaemon on the head node communicates with CMDaemons on the other nodes. The communication includes CMDaemon calls and monitoring data. By default, the management network is set to internalnet for Type 1 and Type 2 networks, and managementnet in Type 3 networks. It is a partition-level cluster-wide setting by default. Partition-level settings can be overridden by the category level setting, and node-level settings can override category level or partition level settings.

- the “Node name” can be set to decide the prefix part of the node name. By default, set to node.

- the “Node digits” can be set to decide the possible size of numbers used for suffix part of the node name. By default, set to 3.

- the “Default category”. This sets the category the nodes are in by default. By default, it is set to default.

- the “Default software image”. This sets the image the nodes use by default, for new nodes that are not in a category yet. By default, it is set to default-image.

- the “Name server”. This sets the name server used by the cluster. By default, it is set to the head node. The default configuration uses the internal network interface and serves only internal hosts. Internal hosts can override using this value at category level (page 73). By default external hosts cannot use this service. To allow external hosts to use the service for resolving internal hosts, the PublicDNS directive (Appendix C) must be set to True.

3. The internal IP addresses and other internal interface values: In Bright View, the network for a node, such as a physical node node001, can be configured via a clickpath Devices→Nodes[node001]→Interface. The subwindow that opens up then allows network configuration to be done for such nodes in a similar way to figure 3.8.

In cmsh the configuration can be done by changing properties from the interfaces submode that is within the device mode for the node.

The items that can be set include:

- the Network device name: By default, this is set to BOOTIF for a node that boots from the same interface as the one from which it is provisioned.
• the Network: By default, this is set to a value of internalnet.

• the IP address: By default, this is automatically assigned a static value, in the range 10.141.0.1 to 10.141.255.255, with the first node being given the first address. Using a static IP address is recommended for reliability, since it allows nodes that lose connectivity to the head node to continue operating. The static address can be changed manually, in case there is an IP address or node ID conflict due to an earlier inappropriate manual intervention. Administrators who want DHCP addressing on the internal network, despite the consequences, can set it via a checkbox.

• onnetworkpriority: This sets the priority of DNS resolution queries for the interface on the network. The range of values that it can take is from 0 to 4294967295. Resolution takes place via the interface with the higher value.

The default priority value for a network interface is set according to its type. These defaults are:

<table>
<thead>
<tr>
<th>Type</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bridge</td>
<td>80</td>
</tr>
<tr>
<td>Bond</td>
<td>70</td>
</tr>
<tr>
<td>Physical</td>
<td>60</td>
</tr>
<tr>
<td>VLAN</td>
<td>50</td>
</tr>
<tr>
<td>Alias</td>
<td>40</td>
</tr>
<tr>
<td>Tunnel</td>
<td>30</td>
</tr>
<tr>
<td>Netmap</td>
<td>20</td>
</tr>
<tr>
<td>BMC</td>
<td>10</td>
</tr>
</tbody>
</table>

• Additional Hostname: In the case of nodes this is in addition to the default node name set during provisioning. The node name set during provisioning takes a default form of node<3 digit number>, as explained earlier on page 71 in the section describing the cluster object settings.

For example, a regular node that has an extra interface, eth1, can have its values set as follows:

Example

```
[bright82] device interfaces node001
[bright82->device[node001]->interfaces]% add physical eth1
[bright82->...->interfaces*[eth1*]]% set network externalnet
[bright82->...->interfaces*[eth1*]]% set additionalhostnames extra01
[bright82->...->interfaces*[eth1*]]% set ip 10.141.1.1
[bright82->...->interfaces*[eth1*]]% commit
[bright82->...->interfaces*[eth1*]]% ..
[bright82->...->interfaces]% ..
[bright82->device[node001]]% reboot
```

4. Node category network values: are settings for the internal network that can be configured for node categories. For example, for the default category called default this can be carried out:

• via Bright View, using the Settings option for Node categories. This is in the clickpath Grouping → Node categories[default-image] → Settings

• or via the category mode in cmsh, for the default node category.
If there are individual node settings that have been configured in Bright View or cmsh, then the node settings override the corresponding category settings for those particular nodes.

The category properties involved in internal networking that can be set include:

- **Default gateway**: The default gateway route for nodes in the node category. If unset, or 0.0.0.0 (the default), then the node default gateway route is decided by the internal network object *Gateway* value. If the default gateway is set as a node category value, then nodes use the node category value as their default gateway route instead.

- **Management network**: The management network is the network used by CMDaemon to manage devices. The default setting is a property of the node object. It can be set as a category property.

- **Name server, Time server, Search domain**: The default setting for these on all nodes is set by the node-installer to refer to the head node, and is not configurable at the node level using Bright View or cmsh. The setting can however be set as a category property, either as a space-separated list of entries or as a single entry, to override the default value.

**Application To Generic Network Devices**: The preceding details for the internal network parameters of the cluster, starting from page 70, are applicable to regular nodes, but they are often also applicable to generic network devices (section 2.1.1). Benefits of adding a generic device to be managed by Bright Cluster Manager include that:

- the name given to the device during addition is automatically placed in the internal DNS zone, so that the device is accessible by name

- the device status is automatically monitored via an ICMP ping (Appendix G.2.1).

- the device can be made to work with the health check and metric framework. The scripts used in the framework will however almost certainly have to be customized to work with the device

After changing network configurations, a reboot of the device is necessary to activate the changes.

**Changing The Global Network Parameters For The Cluster**

The global network `globalnet` is a unique network used to set up a common name space for all nodes in a cluster in Bright Cluster Manager. It is required in Bright Cluster Manager because of the added cloud extension functionality, described in the *Cloudbursting Manual*. Regular nodes and regular cloud nodes are thus both named under the `globalnet` domain name, which is `cm.cluster` by default. So, for example, if default host names for regular nodes (node001, node002, ...) and regular cloud nodes (cnode001, cnode002, ...) are used, node addresses with the domain are:

- `node001.cm.cluster` for node001
- `cnode001.cm.cluster` for cnode001

The only parameter that can be sensibly changed on `globalnet` is the domain name, which is `cm.cluster` by default.

Removing `globalnet` should not be done because it will cause various networking failures, even for clusters deploying no cloud nodes.

Details on how resolution is carried out with the help of `globalnet` are given in section 6.4.1 of the *Cloudbursting Manual*.
Configuring The Cluster

Setting Static Routes In The `staticroutes` Submode Of `cmsh`

To route via a specific gateway, the `staticroutes` submode can be used. This can be set for regular nodes and head nodes via the `device` mode, and for node categories via the `category` mode.

On a newly-installed cluster with a type 1 network (section 3.3.7 of the *Installation Manual*), a node by default routes packets using the head node as the default gateway.

If the administrator would like to configure a regular node to use another gateway to reach a printer on another subnet, as illustrated by figure 3.9:

![Figure 3.9: Example Of A Static Route To A Printer On Another Subnet](image_url)

then an example session for setting the static route is as follows:

**Example**

```
[root@bright82 ~]# cmsh
[bright82 %] device use node001
[bright82->device[node001]]% [bright82->device[node001]]% staticroutes
[bright82->device[node001]->staticroutes]% list
Name (key)          Gateway         Destination
--------------------- ----------------- ---------------
[bright82->device[node001]->staticroutes]% add printerroute
[bright82->...[printerroute*]]% set gateway 10.141.1.1
[bright82->...[printerroute*]]% set destination 192.168.0.3
[bright82->...[printerroute*]]% set networkdevicename bootif
[bright82->...[printerroute*]]% commit
[bright82->...[printerroute]]% show
Parameter          Value
--------------------- ---------------------
Destination        192.168.0.3/32
Gateway            10.141.1.1
Name               printerroute
Network Device Name bootif
```

© Bright Computing, Inc.
3.2 Network Settings

Revision
%
[bright82->device[node001]->staticroutes[printerroute]]% exit
[bright82->device[node001]->staticroutes]% list

Name (key)      Gateway       Destination
----------------- -------------- --------------
printerroute     10.141.1.1    192.168.0.3/32
%
[bright82->device[node001]->staticroutes]% exit; exit
[bright82->device]% reboot node001

In the preceding example, the regular node node001, with IP address 10.141.0.1 can connect to a host 192.168.0.3 outside the regular node subnet using a gateway with the IP addresses 10.141.1.1 and 192.168.0.1. The route is given the arbitrary name printerroute for administrator and CMDaemon convenience, because the host to be reached in this case is a print server. The networkdevicename is given the interface name bootif. If another device interface name is defined in CMDaemon, then that can be used instead. If there is only one interface, then networkdevicename need not be set.

After a reboot of the node, the route that packets from the node follow can be seen with a traceroute or similar. The ping utility with the -R option can often track up to 9 hops, and can be used to track the route:

Example
[
[root@bright82 ~]# ssh node001 ping -c1 -R 192.168.0.3
PING 192.168.0.3 (192.168.0.3) 56(124) bytes of data.
64 bytes from 192.168.0.3: icmp_seq=1 ttl=62 time=1.41 ms
RR: 10.141.0.1
  10.141.1.1
  192.168.0.1
  192.168.0.3
  192.168.0.1
  192.168.0.1
  10.141.1.1
  10.141.0.1
--- 192.168.0.3 ping statistics ---
1 packets transmitted, 1 received, 0% packet loss, time 1ms
rtt min/avg/max/mdev = 1.416/1.416/1.416/0.000 ms
[
[root@bright82->device[node001]->staticroutes]%

The routing is achieved by CMDaemon making sure that whenever the network interface is brought up, the OS of the regular node runs a routing command to configure the routing tables. The command executed for the given example is either:

    route add -host 192.168.0.3 gw 10.141.1.1

or its modern iproute2 equivalent:

    ip route add 192.168.0.3 via 10.141.1.1

If the device bootif is eth0—which is often the case—then the command is run from the network interface configuration file: /etc/sysconfig/network-scripts/ifcfg-eth0 (or /etc/sysconfig/network/ifcfg-eth0 in SUSE).

3.2.4 Tools For Viewing Cluster Connections

Viewing Node Routes With routes In cmsh

The cmsh routes command is a wrapper around the Linux route command, and is designed to run in parallel over nodes. It is intended to get a fast route overview for one or more nodes, and display it for easy comparison.

To get a full overview of the routes for all nodes, the command is run without any options:
Example

[root@bright82 ~]# cmsh
[bright82]% device
[bright82->device]% routes

<table>
<thead>
<tr>
<th>Node</th>
<th>Destination</th>
<th>Gateway</th>
<th>Netmask</th>
<th>Interface</th>
</tr>
</thead>
<tbody>
<tr>
<td>bright82</td>
<td>0.0.0.0</td>
<td>192.168.200.254</td>
<td>0.0.0.0</td>
<td>eth1</td>
</tr>
<tr>
<td>bright82</td>
<td>10.141.0.0</td>
<td>0.0.0.0</td>
<td>255.255.0.0</td>
<td>eth0</td>
</tr>
<tr>
<td>bright82</td>
<td>169.254.0.0</td>
<td>0.0.0.0</td>
<td>255.255.0.0</td>
<td>eth1</td>
</tr>
<tr>
<td>bright82</td>
<td>169.254.169.254</td>
<td>192.168.200.254</td>
<td>255.255.255.255</td>
<td>eth1</td>
</tr>
<tr>
<td>bright82</td>
<td>192.168.200.0</td>
<td>0.0.0.0</td>
<td>255.255.255.0</td>
<td>eth1</td>
</tr>
<tr>
<td>node001</td>
<td>0.0.0.0</td>
<td>10.141.255.254</td>
<td>0.0.0.0</td>
<td>eth0</td>
</tr>
<tr>
<td>node001</td>
<td>10.141.0.0</td>
<td>0.0.0.0</td>
<td>255.255.0.0</td>
<td>eth0</td>
</tr>
<tr>
<td>node001</td>
<td>169.254.0.0</td>
<td>0.0.0.0</td>
<td>255.255.0.0</td>
<td>eth0</td>
</tr>
</tbody>
</table>

To select or filter the output, the grouping options of routes, or the text processing utility awk can be used. Grouping options are options to select nodes, groups, categories, and so on, and are described in the cluster management chapter, on page 37, while a handy link for awk one-liners is http://tuxgraphics.org/~guido/scripts/awk-one-liner.html.

Selection or filtering makes it very easy to detect badly configured nodes. For example, for unexpected gateways, when the expected gateway is `<expected-gateway>`, the following command may be used:

Example

[root@bright82 ~]# cmsh
[bright82]% device
[bright82->device]% routes --category default | awk 'if ($3 != "<expected-gateway>") print $0'

Viewing Connections In cmsh

The connections command can be run from within the device mode of cmsh. It is a parallel wrapper to view active TCP and UDP connections. That is, it runs as a simple command over multiple devices at the same time.

Running connections without options displays a full overview of the currently active TCP and UDP ports on all devices (output truncated):

Example

[bright82->device]% connections

<table>
<thead>
<tr>
<th>Node</th>
<th>Type</th>
<th>Source</th>
<th>Port</th>
<th>Destination</th>
<th>Port</th>
<th>State</th>
</tr>
</thead>
<tbody>
<tr>
<td>bright82</td>
<td>TCP</td>
<td>0.0.0.0</td>
<td>22</td>
<td>0.0.0.0</td>
<td>0</td>
<td>Listening</td>
</tr>
<tr>
<td>bright82</td>
<td>TCP</td>
<td>0.0.0.0</td>
<td>25</td>
<td>0.0.0.0</td>
<td>0</td>
<td>Listening</td>
</tr>
<tr>
<td>bright82</td>
<td>TCP</td>
<td>0.0.0.0</td>
<td>111</td>
<td>0.0.0.0</td>
<td>0</td>
<td>Listening</td>
</tr>
<tr>
<td>bright82</td>
<td>TCP</td>
<td>0.0.0.0</td>
<td>636</td>
<td>0.0.0.0</td>
<td>0</td>
<td>Listening</td>
</tr>
</tbody>
</table>

Filtering with grep can then show which nodes are listening on which ports. For example, nodes listening for DNS queries (port 53) can be found with:

Example

[bright82->device]% connections | head -2; connections | grep Listen | grep " 53"

© Bright Computing, Inc.
and shows that only the head node is listening on that port, providing DNS.

Some third party tools require a free port on all nodes for a service to listen on. Filtering and sorting the output of the connections command allows the administrator to find all the existing used ports across all nodes:

**Example**

```
[bright82->device]# connections | awk 'print $4' | sort -un
```

<table>
<thead>
<tr>
<th>Port</th>
</tr>
</thead>
<tbody>
<tr>
<td>22</td>
</tr>
<tr>
<td>25</td>
</tr>
<tr>
<td>53</td>
</tr>
<tr>
<td>67</td>
</tr>
<tr>
<td>68</td>
</tr>
<tr>
<td>69</td>
</tr>
</tbody>
</table>

Options to the connections command can be seen by running `help connections`. Options include node grouping options (such as `-n|--nodes`, `-c|--category`, and `-t|--type`), and filtering out TCP6 and UDP6 connections (`--no-tcp6`, `--no-udp6`).

**Viewing Connectivity in cmsh**

The connectivity command can be run from within the `device` mode of `cmsh`. It runs ICMP pings along each node route on a network of the cluster. By default the network is the management network.

By default the output displays if the connection is OK, the ping sequence ID (counting starts from zero), and the latency between the source and destination:

**Example**

```
[bright82->device]# connectivity
```

<table>
<thead>
<tr>
<th>Source</th>
<th>Destination</th>
<th>Result</th>
<th>ID</th>
<th>Latency</th>
</tr>
</thead>
<tbody>
<tr>
<td>bright82</td>
<td>node001</td>
<td>Ok</td>
<td>0</td>
<td>0.4ms</td>
</tr>
<tr>
<td>bright82</td>
<td>node002</td>
<td>Ok</td>
<td>0</td>
<td>0.3ms</td>
</tr>
<tr>
<td>bright82</td>
<td>node003</td>
<td>Ok</td>
<td>0</td>
<td>0.4ms</td>
</tr>
<tr>
<td>bright82</td>
<td>bright82</td>
<td>Ok</td>
<td>0</td>
<td>0.1ms</td>
</tr>
</tbody>
</table>

Pings where the source and destination have identical names are carried out via identical interfaces.

Further options to the `connectivity` command can be seen by running `help connectivity`. Options include the ability to set ping timeouts and the cluster network on which to ping, as well as node grouping options.

For larger clusters, the following type of connectivity check may be a helpful diagnostic:

**Example**

```
[bright82->device]# connectivity --statistics --count 100 --delay 0.01
```

<table>
<thead>
<tr>
<th>Name</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>OK</td>
<td>400</td>
</tr>
<tr>
<td>Total</td>
<td>400</td>
</tr>
<tr>
<td>Count</td>
<td>2</td>
</tr>
<tr>
<td>Average</td>
<td>0.35ms</td>
</tr>
<tr>
<td>Minimum</td>
<td>0.35ms</td>
</tr>
<tr>
<td>Maximum</td>
<td>0.35ms</td>
</tr>
<tr>
<td>Uniformity</td>
<td>100.0%</td>
</tr>
</tbody>
</table>

© Bright Computing, Inc.
The preceding shows how uniform nodes are in ping timings. A significant spread in uniformity can indicate network problems.

### 3.3 Configuring Bridge Interfaces

Bridge interfaces can be used to divide one physical network into two separate network segments at layer 2 without creating separate IP subnets. A bridge thus connects the two networks together at layer 3 in a protocol-independent way.

The network device name given to the bridge interface is of the form `br<number>`. The following example demonstrates how to set up a bridge interface in `cmsh`, where the name `br0` is stored by the parameter `networkdevice_name`.

**Example**

```
[bright82->device[node001]->interfaces]% add bridge br0
[bright82->...->interfaces*] % set network internalnet
[bright82->...->interfaces*] % set ip 10.141.1.1
[bright82->...->interfaces*] % show
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Additional Hostnames</td>
<td>no</td>
</tr>
<tr>
<td>DHCP</td>
<td>no</td>
</tr>
<tr>
<td>Forward Delay</td>
<td>0</td>
</tr>
<tr>
<td>IP</td>
<td>10.141.1.1</td>
</tr>
<tr>
<td>Interfaces</td>
<td>00:00:00:00:00:00</td>
</tr>
<tr>
<td>MAC</td>
<td>internalnet</td>
</tr>
<tr>
<td>Network</td>
<td>br0</td>
</tr>
<tr>
<td>Revision</td>
<td>bridge</td>
</tr>
</tbody>
</table>

A bridge interface is composed of one or more physical interfaces. The IP and network fields of the member interfaces must be empty:

**Example**

```
[bright82->...->interfaces*] % set interfaces eth1 eth2; exit
[bright82->...->interfaces*] % clear eth1 ip; clear eth1 network
[bright82->...->interfaces*] % clear eth2 ip; clear eth2 network
[bright82->...->interfaces*] % use br0; commit
```

The `BOOTIF` interface is also a valid interface option.

Currently, the following bridge properties can be set:

- **SpanningTreeProtocol**: sets the spanning tree protocol to be on or off. The default is off.
- **Forward Delay**: sets the delay for forwarding Ethernet frames, in seconds. The default is 0.

Additional properties, if required, can be set manually using the `brctl` command in the OS shell. When listing interfaces in `cmsh`, if an interface is a member of a bond (or bridge) interface, then the corresponding bond or bridge interface name is shown in parentheses after the member interface name:

**Example**

© Bright Computing, Inc.
3.4 Configuring VLAN interfaces

A VLAN (Virtual Local Area Network) is an independent logical LAN within a physical LAN network. VLAN tagging is used to segregate VLANs from each other. VLAN tagging is the practice of inserting a VLAN ID tag into a packet frame header, so that each packet can be associated with a VLAN.

The physical network then typically has sections that are VLAN-aware or VLAN-unaware. The nodes and switches of the VLAN-aware section are configured by the administrator to decide which traffic belongs to which VLAN.

A VLAN interface can be configured for an interface within Bright Cluster Manager using cmsh or Bright View.

3.4.1 Configuring A VLAN Interface Using cmsh

In the following cmsh session, a regular node interface that faces a VLAN-aware switch is made VLAN-aware, and assigned a new interface—an alias interface. It is also assigned a network, and an IP address within that network:

Example

```
[root@bright82 ~]# cmsh
[bright82]# device interfaces node001
[bright82->device[node001]->interfaces]% list
Type Network device name IP Network
----------------- ------------------ -------------- --------------
physical BOOTIF [prov] 10.141.0.1 internalnet
[bright82->device[node001]->interfaces]% list -h | tail -4
Arguments:
type
  alias, bmc, bond, bridge, netmap, physical, tunnel, vlan
[bright82->device[node001]->interfaces]% add vlan BOOTIF.1
[bright82->device[node001]->interfaces][BOOTIF.1]*% commit
[bright82->device[node001]->interfaces][BOOTIF.1]*% show
```

Parameter Value
----------------- --------------
Additional Hostname no
DHCP no
IP 0.0.0.0
MAC 00:00:00:00:00:00

© Bright Computing, Inc.
3.4.2 Configuring A VLAN Interface Using Bright View

Within Bright View a VLAN interface can be configured by selecting the node with that interface, selecting its Settings option, then selecting the Interfaces option. In the Network Interfaces window that opens up, the Network VLAN interface option is selected and a dialog opens up allowing an IP address, network, and other options to be set (figure 3.10).

3.5 Configuring Bonded Interfaces

3.5.1 Adding A Bonded Interface

The Linux bonding driver allows multiple physical network interfaces that have been configured previously (for example, as on page 72) to be bonded as a single logical bond interface. The behavior of such interfaces is determined by their bonding mode. The modes provide facilities such as hot standby or load balancing.

The driver is included by default on head nodes. To configure a non-head node to use a bonded interface, a Linux kernel module called the bonding module must be included in the kernel modules list of the software image of the node. A bonding interface can then be created, and its properties set as follows:

Example

```
[bright82->device[node001]->interfaces]% add bond bond0
[...->device*[node001*]->interfaces*[bond0*]% set network internalnet
[...->device*[node001*]->interfaces*[bond0*]% set ip 10.141.128.1
[...->device*[node001*]->interfaces*[bond0*]% set mode 3
[...->device*[node001*]->interfaces*[bond0*]% set interfaces eth1 eth2
```
Each bonded interface has a unique set of object properties, called bonding options, that specify how the bonding device operates. The bonding options are a string containing one or more options formatted as option=value pairs, with pairs separated from each other by a space character.

A bonded interface also always has a mode associated with it. By default it is set to 0, corresponding to a balanced round-robin packet transmission.

The 7 bonding modes are:

- 0 – balance-rr
- 1 – active-backup
- 2 – balance-xor
- 3 – broadcast
- 4 – 802.3ad
- 5 – balance-tlb
- 6 – balance-alb

Technically, outside of Bright View or cmsh, the bonding mode is just another bonding option specified as part of the options string. However in Bright Cluster Manager the bonding mode value is set up using the dedicated mode property of the bonded interface, for the sake of clarity. To avoid conflict with the value of the mode property, trying to commit a bonding mode value as an option=value pair will fail validation.

### 3.5.2 Single Bonded Interface On A Regular Node

A single bonded interface on a node can be configured and coexist in several ways on nodes with multiple network interfaces. Possibilities and restrictions are illustrated by the following:

- The bonded interface may be made up of two member interfaces, and a third interface outside of the bond could be the boot interface. (The boot interface is the node interface used to PXE boot the node before the kernel loads (section 5.1)).

- The boot interface could itself be a member of the bonded interface. If the boot interface is a member of a bonded interface, then this is the first bonded interface when interfaces are listed as in the example on page 78.

- The bonded interface could be set up as the provisioning interface. However, the provisioning interface cannot itself be a member of a bonded interface. (The provisioning interface is the node’s interface that picks up the image for the node after the initial ramdisk is running. Chapter 5 covers this in more detail).

- A bonded interface can be set up as the provisioning interface, while having a member interface which is used for PXE booting.

### 3.5.3 Multiple Bonded Interface On A Regular Node

A node can also have multiple bonded interfaces configured. Possibilities and restrictions are illustrated by the following:

- Any one of the configured bonded interfaces can be configured as the provisioning interface. However, as already mentioned in the case of single bond interfaces (section 3.5.2), a particular member of a bonded interface cannot be made the provisioning interface.

- When a bonded interface is set as the provisioning interface, then during the node-installer phase of boot, the node-installer brings up the necessary bonded interface along with all its member interfaces so that node provisioning is done over the bonded interface.
3.5.4 Bonded Interfaces On Head Nodes And HA Head Nodes

It is also possible to configure bonded interfaces on head nodes.

For a single head node setup, this is analogous to setting up bonding on regular nodes.

For a high availability (HA) setup (chapter 14), bonding is possible for internalnet as well as for externalnet, but it needs the following extra changes:

- For the bonded interface on internalnet, the shared internal IP alias interface name (the value of networkdevicename, for example, eth0:0 in figure 14.1) for that IP address should be renamed to the bonded alias interface name on internalnet (for example, bond0:0).

- For the bonded interface on externalnet, the shared external IP alias interface name (the value of networkdevicename, for example, eth1:0 in figure 14.1) for that IP address should be renamed to the bonded alias interface name on externalnet (for example, bond1:0).

- Additionally, when using a bonded interface name for the internal network, the value of the provisioning network interface name (the value of provisioninginterface, for example, eth0) for the head nodes, must be changed to the name of the bonded interface (for example, bond0 [prov]) on the internal network. The provisioninginterface value setting is described further on page 185.

Example

```
[headnode1->device[headnode1]->interfaces]% list
Type   Network device name   IP         Network
------- -------------------- -------------- ----
alias   bond0:0                        10.141.255.252 internalnet
alias   bond1:0                        10.150.57.1 externalnet
bond    bond0 [prov]                  10.141.255.254 internalnet
bond    bond1                        10.150.57.3 externalnet
physical eth0 (bond0)                0.0.0.0
physical eth1 (bond0)                0.0.0.0
physical eth2 (bond1)                0.0.0.0
physical eth3 (bond1)                0.0.0.0
```

3.5.5 Tagged VLAN On Top Of a Bonded Interface

It is possible to set up a tagged VLAN interface on top of a bonded interface. There is no requirement for the bonded interface to have an IP address configured in this case. The IP address can be set to 0.0.0.0, however a network must be set.

Example

```
[headnode1->device[node001]->interfaces]% list
Type   Network device name   IP         Network
------- -------------------- -------------- ----
bond    bond0                        0.0.0.0 internalnet
physical eth0 [prov]                10.141.0.1 internalnet
physical eth1 (bond0)               0.0.0.0
physical eth2 (bond0)               0.0.0.0
vlan    bond0.3                      10.150.1.1 othernet
```

3.5.6 Further Notes On Bonding

If using bonding to provide failover services, miimon, which is off by default (set to 0), should be given a value. The miimon setting is the time period in milliseconds between checks of the interface carrier state. A common value is miimon=100.

When listing interfaces in cmsh, if an interface is a member of a bond or bridge interface, then the corresponding bonded or bridge interface name is shown in parentheses after the member interface.
name. Section 3.3, on configuring bridge interfaces, shows an example of such a listing from within cmsh on page 78.

More on bonded interfaces (including a detailed description of bonding options and modes) can be found at http://www.kernel.org/doc/Documentation/networking/bonding.txt.

3.6 Configuring InfiniBand And Omni-Path Interfaces

On clusters with an InfiniBand interconnect, the InfiniBand Host Channel Adapter (HCA) in each node must be configured before it can be used. A similar requirement applies for Omni-Path and its fabric controller. Bright Cluster Manager manages InfiniBand and Omni-Path configuration in the same way. Therefore Omni-Path management is usually also implied whenever InfiniBand is discussed in the Bright Cluster Manager manuals.

This section describes how to set up the InfiniBand service on the nodes for regular use. Setting up InfiniBand for booting and provisioning purposes is described in Chapter 5, while setting up InfiniBand for NFS is described in section 3.10.5.

3.6.1 Installing Software Packages

On a standard Bright Cluster Manager cluster, the OFED (OpenFabrics Enterprise Distribution) packages that are part of the Linux base distribution are used. These packages provide RDMA implementations allowing high bandwidth/low latency interconnects on OFED hardware. The implementations can be used by InfiniBand hardware, and iWarp protocol hardware such as the hardware-accelerated RDMA over ethernet provided by Intel.

By default, all relevant OFED packages are installed on the head node and software images. It is possible to replace the distribution OFED with an OFED provided by the Bright Cluster Manager repository or another custom version. The replacement can be for the entire cluster, or only for certain software images. Administrators may choose to switch to a different OFED version if the HCAs used are not supported by the distribution OFED version, or to increase performance by using an OFED version that has been optimized for a particular HCA. Installing the Bright Cluster Manager OFED and Omni-Path packages is covered in section 7.6 and section 7.7 respectively of the Installation Manual.

If the InfiniBand network is enabled during installation, then the rdma script runs during the init stage of booting up for the enabled nodes. For SLES and Linux distributions based on versions prior to Red Hat 6, the openibd script is used instead of the rdma script.

The rdma or openibd script takes care of loading the relevant InfiniBand HCA kernel modules. When adding an InfiniBand network after installation, it may be necessary to use chkconfig manually to configure the rdma or openibd script to be run at boot-time on the head node and inside the software images.

3.6.2 Subnet Managers

Every InfiniBand subnet requires at least one subnet manager to be running. The subnet manager takes care of routing, addressing and initialization on the InfiniBand fabric. Some InfiniBand switches include subnet managers. However, on large InfiniBand networks or in the absence of a switch-hosted subnet manager, a subnet manager needs to be started on at least one node inside of the cluster. When multiple subnet managers are started on the same InfiniBand subnet, one instance will become the active subnet manager whereas the other instances will remain in passive mode. It is recommended to run 2 subnet managers on all InfiniBand subnets to provide redundancy in case of failure.

On a Linux machine that is not running Bright Cluster Manager, an administrator sets a subnet manager service\(^1\) to start at boot-time with a command such as: “chkconfig opensm on”. However, for clusters managed by Bright Cluster Manager, a subnet manager is best set up using CMDaemon. There are two ways of setting CMDaemon to start up the subnet manager on a node at boot time:

1. by assigning a role.

\(^1\)usually opensm, but opensmd in SLES
In cmsh this can be done with:

```
[root@bright82 ~]# cmsh -c "device roles <node>; assign subnetmanager; commit"
```

where `<node>` is the name of a node on which it will run, for example: bright82, node001, node002...

In Bright View, the subnet manager role is assigned by selecting a head node or regular node from the Devices resource, and assigning it the "Subnet manager role". The clickpath for this, for a node node002 for example, is Devices→Nodes→node002→Settings→Roles→Add[Subnet manager role].

2. by setting the service up. Services are covered more generally in section 3.11.

In cmsh this is done with:

**Example**

```
[root@bright82 ~]# cmsh
[bright82]# device services node001
[bright82->device[node001]->services]% add opensm
[bright82->device[node001]->services[opensm]% set autostart yes
[bright82->device[node001]->services[opensm]% set monitored yes
[bright82->device[node001]->services[opensm]% commit
[bright82->device[node001]->services[opensm]]%
```

In Bright View the subnet manager service is configured by selecting a head node or regular node from the resources tree, and adding the service to it. The clickpath for this, for a node node002 for example, is: Devices→Nodes→node002→Settings→Services[Service].

When the head node in a cluster is equipped with an InfiniBand HCA, it is a good candidate to run as a subnet manager for smaller clusters.

On large clusters a dedicated node is recommended to run the subnet manager.

### 3.6.3 InfiniBand Network Settings

Although not strictly necessary, it is recommended that InfiniBand interfaces are assigned an IP address (i.e. IP over IB). First, a network object in the cluster management infrastructure should be created. The procedure for adding a network is described in section 3.2.2. The following settings are recommended as defaults:

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>ibnet</td>
</tr>
<tr>
<td>Domain name</td>
<td>ib.cluster</td>
</tr>
<tr>
<td>Type</td>
<td>internal</td>
</tr>
<tr>
<td>Base address</td>
<td>10.149.0.0</td>
</tr>
<tr>
<td>Netmask bits</td>
<td>16</td>
</tr>
<tr>
<td>MTU</td>
<td>up to 4k in datagram mode</td>
</tr>
<tr>
<td></td>
<td>up to 64k in connected mode</td>
</tr>
</tbody>
</table>

By default, an InfiniBand interface is set to datagram mode, because it scales better than connected mode. It can be configured to run in connected mode by setting the connectedmode property:

**Example**
3.6 Configuring InfiniBand And Omni-Path Interfaces

For nodes that are PXE booting or are getting provisioned over InfiniBand, the mode setting in the node-installer script has to be changed accordingly.

Example

```
[root@bright82 ~]# echo datagram > /cm/node-installer/scripts/ipoib_mode
```

Once the network has been created all nodes must be assigned an InfiniBand interface on this network. The easiest method of doing this is to create the interface for one node device and then to clone that device several times.

For large clusters, a labor-saving way to do this is using the addinterface command (section 3.7.1) as follows:

```
[root@bright82 ~]# echo "device addinterface -n node001..node150 physical ib0 ibnet 10.149.0.1 commit" | cmsh -x
```

When the head node is also equipped with an InfiniBand HCA, it is important that a corresponding interface is added and configured in the cluster management infrastructure.

Example

Assigning an IP address on the InfiniBand network to the head node:

```
[bright82->device[bright82]->interfaces]% add physical ib0
[bright82->device[bright82]->interfaces*[ib0*]]% set network ibnet
[bright82->device[bright82]->interfaces*[ib0*]]% set ip 10.149.255.254
[bright82->device[bright82]->interfaces*[ib0*]]% commit
```

As with any change to the network setup, the head node needs to be restarted to make the above change active.

3.6.4 Verifying Connectivity

After all nodes have been restarted, the easiest way to verify connectivity is to use the ping utility

Example

Pinging node015 while logged in to node014 through the InfiniBand interconnect:

```
[root@node014 ~]# ping node015.ib.cluster
PING node015.ib.cluster (10.149.0.15) 56(84) bytes of data.
64 bytes from node015.ib.cluster (10.149.0.15): icmp_seq=1 ttl=64 time=0.086 ms
... ...
```

If the ping utility reports that ping replies are being received, the InfiniBand is operational. The ping utility is not intended to benchmark high speed interconnects. For this reason it is usually a good idea to perform more elaborate testing to verify that bandwidth and latency are within the expected range.

The quickest way to stress-test the InfiniBand interconnect is to use the Intel MPI Benchmark (IMB), which is installed by default in `/cm/shared/apps/imb/current`. The `setup.sh` script in this directory can be used to create a template in a user’s home directory to start a run.

Example

Running the Intel MPI Benchmark using openmpi to evaluate performance of the InfiniBand interconnect between node001 and node002:
Configuring The Cluster

```bash
[root@bright82 ~]# su - cmsupport
[cmsupport@bright82 ~]$ cd /cm/shared/apps/imb/current/
[cmsupport@bright82 current]$ ./setup.sh
[cmsupport@bright82 current]$ cd ~/BenchMarks/imb/2017
[cmsupport@bright82 2017]$ module load openmpi/gcc
[cmsupport@bright82 2017]$ module initadd openmpi/gcc
[cmsupport@bright82 2017]$ make -f make_mpi2
[cmsupport@bright82 2017]$ mpirun -np 2 -machinefile ../nodes IMB-MPI1 PingPong

#---------------------------------------------------
# Benchmarking PingPong
# #processes = 2
#---------------------------------------------------
```

<table>
<thead>
<tr>
<th>#bytes</th>
<th>#repetitions</th>
<th>t[usec]</th>
<th>Mbytes/sec</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0</td>
<td>1.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>1</td>
<td>1.08</td>
<td>0.88</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>1.07</td>
<td>1.78</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>1.08</td>
<td>3.53</td>
</tr>
<tr>
<td></td>
<td>8</td>
<td>1.08</td>
<td>7.06</td>
</tr>
<tr>
<td></td>
<td>16</td>
<td>1.16</td>
<td>13.16</td>
</tr>
<tr>
<td></td>
<td>32</td>
<td>1.17</td>
<td>26.15</td>
</tr>
<tr>
<td></td>
<td>64</td>
<td>1.17</td>
<td>52.12</td>
</tr>
<tr>
<td></td>
<td>128</td>
<td>1.20</td>
<td>101.39</td>
</tr>
<tr>
<td></td>
<td>256</td>
<td>1.37</td>
<td>177.62</td>
</tr>
<tr>
<td></td>
<td>512</td>
<td>1.69</td>
<td>288.67</td>
</tr>
<tr>
<td></td>
<td>1024</td>
<td>2.30</td>
<td>425.34</td>
</tr>
<tr>
<td></td>
<td>2048</td>
<td>3.46</td>
<td>564.73</td>
</tr>
<tr>
<td></td>
<td>4096</td>
<td>7.37</td>
<td>530.30</td>
</tr>
<tr>
<td></td>
<td>8192</td>
<td>11.21</td>
<td>697.20</td>
</tr>
<tr>
<td></td>
<td>16384</td>
<td>21.63</td>
<td>722.24</td>
</tr>
<tr>
<td></td>
<td>32768</td>
<td>42.19</td>
<td>740.72</td>
</tr>
<tr>
<td></td>
<td>65536</td>
<td>70.09</td>
<td>891.69</td>
</tr>
<tr>
<td></td>
<td>131072</td>
<td>125.46</td>
<td>996.35</td>
</tr>
<tr>
<td></td>
<td>262144</td>
<td>238.04</td>
<td>1050.25</td>
</tr>
<tr>
<td></td>
<td>524288</td>
<td>500.76</td>
<td>998.48</td>
</tr>
<tr>
<td></td>
<td>1048576</td>
<td>1065.28</td>
<td>938.72</td>
</tr>
<tr>
<td></td>
<td>2097152</td>
<td>2033.13</td>
<td>983.71</td>
</tr>
<tr>
<td></td>
<td>4194304</td>
<td>3887.00</td>
<td>1029.07</td>
</tr>
</tbody>
</table>

# All processes entering MPI_Finalize

To run on nodes other than node001 and node002, the ../nodes file must be modified to contain different hostnames. To perform other benchmarks, the PingPong argument should be omitted.

### 3.7 Configuring BMC (IPMI/iLO/DRAC/CIMC/Redfish) Interfaces

Bright Cluster Manager can initialize and configure the baseboard management controller (BMC) that may be present on devices. This ability can be set during the installation on the head node (section 3.3.8 of the Installation Manual), or it can be set after installation as described in this section. The IPMI, iLO, DRAC, CIMC, or Redfish interface that is exposed by a BMC is treated in the cluster management infrastructure as a special type of network interface belonging to a device. In the most common setup a dedicated network (i.e. IP subnet) is created for BMC communication. The 10.148.0.0/16 network is used by default for BMC interfaces by Bright Cluster Manager.
### 3.7 Configuring BMC (IPMI/iLO/DRAC/CIMC/Redfish) Interfaces

#### 3.7.1 BMC Network Settings

The first step in setting up a BMC is to add the BMC network as a network object in the cluster management infrastructure. The procedure for adding a network is described in section 3.2.2. The following settings are recommended as defaults:

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>bmcnet, ilonet, ipminet, dracnet, cimcnet, or rfnet</td>
</tr>
<tr>
<td>Domain name</td>
<td>bmc.cluster, ilo.cluster, ipmi.cluster, drac.cluster, cimc.cluster, or rf.cluster</td>
</tr>
<tr>
<td>External network</td>
<td>false</td>
</tr>
<tr>
<td>Base address</td>
<td>10.148.0.0</td>
</tr>
<tr>
<td>Netmask bits</td>
<td>16</td>
</tr>
<tr>
<td>Broadcast address</td>
<td>10.148.255.255</td>
</tr>
</tbody>
</table>

Once the network has been created, all nodes must be assigned a BMC interface, of type `bmc`, on this network. The easiest method of doing this is to create the interface for one node device and then to clone that device several times.

For larger clusters this can be laborious, and a simple bash loop can be used to do the job instead:

```
[bright82 ~]# for ((i=1; i<=150; i++)) do
echo "
device interfaces node$(printf '%03d' $i) add bmc ipmi0
set network bmcnet
set ip 10.148.0.$i
commit"; done | cmsh -x  # -x usefully echoes what is piped into cmsh
```

The preceding loop can conveniently be replaced with the `addinterface` command, run from within the device mode of `cmsh`:

```
[bright82 ~]# echo "
device
addinterface -n node001..node150 bmc ipmi0 bmcnet 10.148.0.1
commit" | cmsh -x
```

The help text in cmsh gives more details on how to use `addinterface`.

Most administrators are likely to simply run it as an interactive session in cmsh, running the help `addinterface` command for reference, and then supplying the options for the nodes and interface settings in device mode. For example, as in the following session:

```
[root@bright82 ~]# cmsh
[bright82]# device
[bright82->device]# help addinterface
Name: addinterface - Add a network interface to one or more nodes

Usage: addinterface [OPTIONS] <type> <devicename> <network> <firstip>

Options: ...

Examples: addinterface -n node001..node010 physical ib0 ibnet 10.149.0.1
```

© Bright Computing, Inc.
In order to be able to communicate with the BMC interfaces, the head node also needs an interface on the BMC network. Depending on how the BMC interfaces are physically connected to the head node, the head node has to be assigned an IP address on the BMC network one way or another. There are two possibilities for how the BMC interface is physically connected:

- When the BMC interface is connected to the primary internal network, the head node should be assigned an alias interface configured with an IP address on the BMC network.
- When the BMC interface is connected to a dedicated physical network, the head node must also be physically connected to this network. A physical interface must be added and configured with an IP address on the BMC network.

**Example**

Assigning an IP address on the BMC network to the head node using an alias interface:

```
[bright82->device->interfaces]% add alias eth0:0
[bright82->device->interfaces+[eth0:0+]]% set network bmcnet
[bright82->device->interfaces+[eth0:0+]]% set ip 10.148.255.254
[bright82->device->interfaces+[eth0:0+]]% commit
```

As with any change to the network setup, the head node needs to be restarted to make the above change active, although in this particular case restarting the network service would suffice.

BMC connectivity from the head node to the IP addresses of the configured interfaces on the regular nodes can be tested with Bash one-liner such as:

```
[root@bright82 ~]# for i in $(cmsh -c "device; foreach -t physicalnode (interfaces; / use ilo0; get ip)"; do ping -c1 $i; done | grep packet
1 packets transmitted, 0 received, 100% packet loss, time 0ms
1 packets transmitted, 0 received, 100% packet loss, time 0ms
1 packets transmitted, 0 received, 100% packet loss, time 0ms
...
```

In the preceding example the packet loss demonstrates there is a connection problem between the head node and the BMC subnet.

### 3.7.2 BMC Authentication

The node-installer described in Chapter 5 is responsible for the initialization and configuration of the BMC interface of a device. In addition to a number of network-related settings, the node-installer also configures BMC authentication credentials. By default BMC interfaces are configured with username `bright` and a random password that is generated during the installation of the head node. The password is stored by CMDaemon. It can be managed from `cmsh` from within the `base` object of `partition` mode, in the `bmcsettings` submode. This means that by default, each BMC in the cluster has that username and password set during node boot.

For example, the current values of the BMC username and password for the entire cluster can be obtained and changed as follows:
Example

[bright82]% partition use base
[bright82->partition[base]]% bmcsettings
[bright82->partition[base]->bmcsettings]% get username
bright
[bright82->partition[base]->bmcsettings]% get password
Za4ohniohMa2zew
[bright82->partition[base]->bmcsettings]% set username bmcadmin
[bright82->partition*[base*]->bmcsettings*]% set password
enter new password: ********
retype new password: ********
[bright82->partition*[base*]->bmcsettings*]% commit

In Bright View, selecting the cluster item in the resources pane, and then using the Settings option, allows the BMC settings to be edited.

It is possible to set the BMC authentication credentials cluster-wide, category, or per node. As usual, category settings override cluster-wide settings, and node settings override category settings. The relevant properties are:

<table>
<thead>
<tr>
<th>Property</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>BMC User ID</td>
<td>User type. Normally set to 4 for administrator access.</td>
</tr>
<tr>
<td>BMC User Name</td>
<td>User name</td>
</tr>
<tr>
<td>BMC Password</td>
<td>Password for specified user name</td>
</tr>
</tbody>
</table>

The Bright Cluster Manager stores the configured BMC username and password. These are used:

- to configure the BMC interface from the node-installer
- to authenticate to the BMC interface after it has come up

BMC management operations, such as power cycling nodes and collecting hardware metrics, can then be performed.

If authentication fails, then an explanation for why can often be found in the node-installer log (/var/log/node-installer).

### 3.7.3 Interfaces Settings

#### Interface Name

It is recommended that the network device name of a BMC interface start with ipmi, ilo, drac, cimc, or rf, according to whether the BMC is running with IPMI, iLO, DRAC, CIMC, or Redfish. Numbers are appended to the base name, resulting in, for example: ipmi0.

#### Obtaining The IP address

BMC interfaces can have their IP addresses configured statically, or obtained from a DHCP server.

Only a node with a static BMC IP address has BMC power management done by Bright Cluster Manager. A node with a DHCP-assigned BMC IP address, requires custom BMC power management (section 4.1.4) due to its dynamic nature.

#### Dell racadm Installation

If Dell was chosen as a hardware option when the Bright Cluster Manager ISO was created for installation, then the Dell OpenManage utilities are located under /opt/dell on the head node.

If Dell is chosen as a hardware option when nodes are configured in the Bright Cluster Manager installer, then the default software image that is used by the node has the Dell OpenManage utilities, located on the head node at /cm/images/default-image/opt/dell.
The Dell OpenManage utilities contain the `racadm` binary, which can be used to issue power commands (Chapter 4). Bright Cluster Manager runs commands similar to the following to issue the power commands:

```
/opt/dell/srvadmin/sbin/racadm -r <DRAC interface IP address> -u <bmcusername> -p <bmcpassword>
  serveraction powerstatus
```

```
/opt/dell/srvadmin/sbin/racadm -r <DRAC interface IP address> -u <bmcusername> -p <bmcpassword>
  serveraction hardreset
```

The BMC username/password values can be obtained from `cmsh` as follows:

```
[root@bright82 ~]# cmsh
[bright82]$ partition use base
[bright82->partition[base]]% bmcsettings
[bright82->partition[base]->bmcsettings]% get password
12345
[bright82->partition[base]->bmcsettings]% get username
tom
```

Sometimes the bright user does not have the right privilege to get the correct values. The `racadm` commands then fail.

The bright user privilege can be raised using the following command:

```
/opt/dell/srvadmin/sbin/racadm -r <DRAC interface IP address> -u root
  -p <root password> set iDRAC.Users.4.Privilege 511
```

Here it is assumed that the bright user has a userID 4, and the privilege can be set to 511.

### 3.7.4 Identification With A BMC

Sometimes it is useful to identify a node using BMC commands. This can be done by, for example, blinking a light via a BMC command on the node:

#### Example

```
ipmitool -U <bmcusername> -P <bmcpassword> -H <host IP> chassis identify 1
```

The exact implementation may be vendor-dependent, and need not be an `ipmitool` command. Such commands can be scripted and run from CMDaemon.

For testing without a BMC, the example script at `/cm/local/examples/cmd/bmc_identify` can be used if the environment variable `$CMD_HOSTNAME` is set. The logical structure of the script can be used as a basis for carrying out an identification task when a physical BMC is in place, by customizing the script and then placing the script in a directory for use.

To have such a custom BMC script run from CMDaemon, the `BMCIdentifyScript` advanced configuration directive (page 643) can be used.

### 3.8 Configuring Switches And PDUs

#### 3.8.1 Configuring With The Manufacturer’s Configuration Interface

Network switches and PDUs that will be used as part of the cluster should be configured with the PDU/switch configuration interface described in the PDU/switch documentation supplied by the manufacturer. Typically the interface is accessed by connecting via a web browser or telnet to an IP address preset by the manufacturer.

The IP address settings of the PDU/switch must match the settings of the device as stored by the cluster manager.
3.8 Configuring Switches And PDUs

- In Bright View, this is done by selecting the Switches resource, selecting the particular switch from within that resource (or adding it via the Add button), then filling in the values in the associated Settings window that comes up (figure 3.11). The IP address can then be set and saved.

- In cmsh this can be done in device mode, with a set command:

  Example

  
  [root@bright82 ~]# cmsh
  [bright82]# device
  [bright82->device]# set switch01 ip 10.141.253.2
  [bright82->device*]# commit

3.8.2 Configuring SNMP

Moreover, in order to allow the cluster management software to communicate with the switch or PDU, SNMP must be enabled and the SNMP community strings should be configured correctly. By default, the SNMP community strings for switches and PDUs are set to public and private for respectively read and write access. If different SNMP community strings have been configured in the switch or PDU, the readstring and writestring properties of the corresponding switch device should be changed.

Example

  [bright82]# device use switch01
  [bright82->device[switch01]]# get readstring
  public
  [bright82->device[switch01]]# get writestring
  private
  [bright82->device[switch01]]# set readstring public2
  [bright82->device*[switch01*]]# set writestring private2
  [bright82->device*[switch01*]]# commit

  Alternatively, these properties can also be set via Bright View from the Switches resource, selecting the switch, and then modifying the string settings by accessing them using the Edit button.

3.8.3 Uplink Ports

Uplink ports are switch ports that are connected to other switches. CMDaemon must be told about any switch ports that are uplink ports, or the traffic passing through an uplink port will lead to mistakes in what CMDaemon knows about port and MAC correspondence. Uplink ports are thus ports that CMDaemon is told to ignore.

To inform CMDaemon about what ports are uplink ports, Bright View or cmsh are used:

- In Bright View, the switch is selected from the Switches resource option folder, and the Edit option opens up a settings window for that switch (figure 3.11). The port number corresponding to uplink port number is filled in the blank field beside the “Uplink:” label. More uplinks can be appended by clicking on the + widget. The state is saved with the Save button.
In cmsh, the switch is accessed from the device mode. The uplink port numbers can be appended one-by-one with the append command, or set in one go by using space-separated numbers.

**Example**

```
[root@bright82 ~]# cmsh
[bright82] % device
[bright82->device] % set switch01 uplinks 15 16
[bright82->device*] % set switch02 uplinks 01
[bright82->device*] % commit
successfully committed 3 Devices
```

### 3.8.4 The showport MAC Address to Port Matching Tool

The showport command can be used in troubleshooting network topology issues, as well as checking and setting up new nodes (section 5.4.2).

**Basic Use Of showport**

In the device mode of cmsh is the showport command, which works out which ports on which switch are associated with a specified MAC address.

**Example**

```
[root@bright82 ~]# cmsh
[bright82] % device
[bright82->device] % showport 00:30:48:30:73:92
[bright82->device] % switch01:12
```

When running showport, CMDaemon on the head node queries all switches until a match is found. If a switch is also specified using the "-s" option, then the query is carried out for that switch first. Thus the preceding example can also be specified as:

```
[bright82->device] % showport -s switch01 00:30:48:30:73:92
[bright82->device] % switch01:12
```

If there is no port number returned for the specified switch, then the scan continues on other switches.
### 3.8 Configuring Switches And PDUs

#### Mapping All Port Connections In The Cluster With `showport`
A list indicating the port connections and switches for all connected devices that are up can be generated using this script:

**Example**

```bash
#!/bin/bash
for nodename in $(cmsh -c "device; foreach * (get hostname)")
do
    macad=$(cmsh -c "device use $nodename; get mac")
    echo -n "$macad $nodename 
    cmsh -c "device showport $macad"
done
```

The script may take a while to finish its run. It gives an output like:

**Example**

```
00:00:00:00:00:00 switch01: No ethernet switch found connected to this mac address
00:30:48:30:73:92 bright82: switch01:12
00:26:6C:F2:AD:54 node001: switch01:1
00:00:00:00:00:00 node002: No ethernet switch found connected to this mac address
```

#### 3.8.5 Disabling Port Detection
An administrator may wish to disable node identification based on port detection. For example, in the case of switches with buggy firmware, the administrator may feel more comfortable relying on MAC-based identification. Disabling port detection can be carried out by clearing the `ethernetswitch` setting of a node, a category, or a group. For example, in `cmsh`, for a node:

**Example**

```
[root@bright82 ~]# cmsh
[bright82]% device use node001
[bright82->device[node001]]% clear ethernetswitch
[bright82->device[node001]]% commit
[bright82->device[node001]]%
```

Or, for example for the default category, with the help of the `foreach` command:

**Example**

```
[root@bright82 ~]# cmsh
[bright82]% device
[bright82->device]]% foreach -c default (clear ethernetswitch); commit
```

#### 3.8.6 The `switchoverview` Command
Also within device mode, the `switchoverview` command gives an overview of MAC addresses detected on the ports, and some related properties. The command works using SNMP queries. Output is similar to the following (some lines ellipsized):

```
[bright82->device]% switchoverview dell-switch1
Device: dell-switch1
State: [ UP ]
Model: 24G Ethernet Switch
```

© Bright Computing, Inc.
### Configuring The Cluster

<table>
<thead>
<tr>
<th>Port</th>
<th>Status</th>
<th>Assigned</th>
<th>Uplink Speed</th>
<th>Detected</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>UP</td>
<td>false</td>
<td>1 Gb/s</td>
<td>-</td>
</tr>
<tr>
<td>2</td>
<td>UP</td>
<td>false</td>
<td>1 Gb/s</td>
<td>-</td>
</tr>
<tr>
<td>3</td>
<td>UP</td>
<td>false</td>
<td>1 Gb/s</td>
<td>74:86:7A:AD:3F:2F, node3</td>
</tr>
<tr>
<td>4</td>
<td>UP</td>
<td>false</td>
<td>1 Gb/s</td>
<td>74:86:7A:AD:43:E9, node4</td>
</tr>
<tr>
<td>...</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>UP</td>
<td>false</td>
<td>1 Gb/s</td>
<td>74:86:7A:AD:44:D8, node11</td>
</tr>
<tr>
<td>...</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>49</td>
<td>UP</td>
<td>false</td>
<td>10 Gb/s</td>
<td>74:86:7A:AD:61:4D, node1</td>
</tr>
<tr>
<td>50</td>
<td>UP</td>
<td>false</td>
<td>10 Gb/s</td>
<td>74:86:7A:AD:41:A0, node2</td>
</tr>
<tr>
<td>53</td>
<td>UP</td>
<td>node34</td>
<td>false</td>
<td>5C:F9:DD:5F:78:3D, node34</td>
</tr>
<tr>
<td>54</td>
<td>UP</td>
<td>node35</td>
<td>false</td>
<td>5C:F9:DD:5F:45:AC, node35</td>
</tr>
<tr>
<td>...</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>180</td>
<td>UP</td>
<td>true</td>
<td>100 Mb/s</td>
<td>QDR-switch</td>
</tr>
<tr>
<td>205</td>
<td>UP</td>
<td>true</td>
<td>10 Gb/s</td>
<td>-</td>
</tr>
<tr>
<td>206</td>
<td>DOWN</td>
<td>false</td>
<td>10 Gb/s</td>
<td>-</td>
</tr>
<tr>
<td>...</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

#### 3.9 Disk Layouts: Disked, Semi-Diskless, And Diskless Node Configuration

Configuring the disk layout for head and regular nodes is done as part of the initial setup (section 3.3.17 of the Installation Manual). For regular nodes, the disk layout can also be re-configured by Bright Cluster Manager once the cluster is running. For a head node, however, the disk layout cannot be re-configured after installation by Bright Cluster Manager, and head node disk layout reconfiguration must then therefore be treated as a regular Linux system administration task, typically involving backups and resizing partitions.

The remaining parts of this section on disk layouts therefore concern regular nodes, not head nodes.

#### 3.9.1 Disk Layouts

A disk layout is specified using an XML schema (Appendix D.1). The disk layout typically specifies the devices to be used, its partitioning scheme, and mount points. Possible disk layouts include the following:

- Default layout (Appendix D.3)
- Hardware RAID setup (Appendix D.4)
- Software RAID setup (Appendix D.5)
- LVM setup (Appendix D.7)
- Diskless setup (Appendix D.9)
- Semi-diskless setup (Appendix D.10)

#### 3.9.2 Disk Layout Assertions

Disk layouts can be set to `assert`
that particular hardware be used, using XML element tags such as vendor or requiredSize (Appendix D.11)

- custom assertions using an XML assert element tag to run scripts placed in CDATA sections (Appendix D.12)

### 3.9.3 Changing Disk Layouts

A disk layout applied to a category of nodes is inherited by default by the nodes in that category. A disk layout that is then applied to an individual node within that category overrides the category setting. This is an example of the standard behavior for categories, as mentioned in section 2.1.3.

By default, the cluster is configured with a standard layout specified in section D.3. The layouts can be accessed from Bright View or cmsh, as is illustrated by the example in section 3.9.4, which covers changing a node from disked to diskless mode:

### 3.9.4 Changing A Disk Layout From Disked To Diskless

The XML schema for a node configured for diskless operation is shown in Appendix D.9. This can often be deployed as is, or it can be modified during deployment using Bright View or cmsh as follows:

#### Changing A Disk Layout Using Bright View

To change a disk layout with Bright View, the current disk layout is accessed by selecting a node category or a specific node from the resource tree. In the Settings pane, the Disk setup field can be edited. Clicking on the Browse button shows several possible configurations that can be loaded up, and if desired, edited to suit the situation (figure 3.12). To switch from the existing disk layout to a diskless one, the diskless XML configuration is loaded and saved to the node or node category.

#### Changing A Disk Layout Using cmsh

To edit an existing disk layout from within cmsh, the existing XML configuration is accessed by editing the disksetup property in device mode for a particular node, or by editing the disksetup property in category mode for a particular category. Editing is done using the set command, which opens up a text editor:

**Example**

```
[root@bright82 ~]# cmsh
[bright82]% device use node001
[bright82->device[node001]]% set disksetup
```

After editing and saving the XML configuration, the change is then committed to CMDaemon with the commit command. It should be understood that a disk layout XML configuration is not stored in a

© Bright Computing, Inc.
file on the filesystem, but in the CMDaemon database. The XML configurations that exist on a default cluster at
/cm/images/default-image/cm/local/apps/cmd/etc/htdocs/disk-setup/
and
/cm/local/apps/cmd/etc/htdocs/disk-setup/
avoid being default configurations.

If the disksetup setting for a device is deleted, using the clear command, then the category level disksetup property is used by the device. This is in accordance with the usual behavior for node values that override category values (section 2.1.5).

Instead of editing an existing disk layout, another XML configuration can also be assigned. A diskless configuration may be chosen and set as follows:

**Example**

[bright82->device[node001]]% set disksetup /cm/local/apps/cmd/\etc/htdocs/disk-setup/slave-diskless.xml

In these preceding Bright View and cmsh examples, after committing the change and rebooting the node, the node then functions entirely from its RAM, without using its own disk.

However, RAM is usually a scarce resource, so administrators often wish to optimize diskless nodes by freeing up the RAM on them from the OS that is using the RAM. Freeing up RAM can be accomplished by providing parts of the filesystem on the diskless node via NFS from the head node. That is, mounting the regular node with filesystems exported via NFS from the head node. The details of how to do this are a part of section 3.10, which covers the configuration of NFS exports and mounts in general.

### 3.10 Configuring NFS Volume Exports And Mounts

NFS allows Unix NFS clients shared access to a filesystem on an NFS server. The accessed filesystem is called an NFS volume by remote machines. The NFS server exports the filesystem to selected hosts or networks, and the clients can then mount the exported volume locally.

An unformatted filesystem cannot be used. The drive must be partitioned beforehand with fdisk or similar partitioning tools, and its filesystem formatted with mkfs or similar before it can be exported.

In Bright Cluster Manager, the head node is typically used to export an NFS volume to the regular nodes, and the regular nodes then mount the volume locally.

- For RHEL6, the client module and server module for NFS over RDMA are provided by default as a combined kernel module. The module is built with the compile option:
  
  - CONFIG_SUNRPC_XPRT_RDMA

- In RHEL7 only the client module is provided by default. The module is built with the compile option:

  - CONFIG_SUNRPC_XPRT_RDMA_CLIENT

  The server module is not provided by default, and must be compiled by the administrator explicitly for the NFS server node. The module can be built by using the compile option:

  - CONFIG_SUNRPC_XPRT_RDMA_SERVER

  However, this is currently (January 2015) unstable and unsupported by Red Hat. The server module may be useable for custom-built kernel versions beyond 3.17. The Bright Computing suggestion for an administrator who needs a very fast network filesystem, is Lustre (section 7.8 of the Installation Manual) over InfiniBand.
If auto-mounting is used, then the configuration files for exporting should be set up on the NFS server, and the mount configurations set up on the software images. The service “autofs” or the equivalent can be set up using Bright View via the “Services” option (section 3.11) on the head and regular nodes or node categories. With cmsh the procedure to configure auto-mounting on the head and regular nodes could be:

**Example**

```bash
[root@bright82 ~]# cmsh
[bright82]# device use bright82
[bright82->device[bright82]]% services
[bright82->device[bright82]->services]% add autofs
[bright82->device*[bright82*]->services*[autofs*]]% show

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Autostart</td>
<td>no</td>
</tr>
<tr>
<td>Belongs to role</td>
<td>no</td>
</tr>
<tr>
<td>Monitored</td>
<td>no</td>
</tr>
<tr>
<td>Revision</td>
<td></td>
</tr>
<tr>
<td>Run if</td>
<td>ALWAYS</td>
</tr>
<tr>
<td>Service</td>
<td>autofs</td>
</tr>
<tr>
<td>Sickness check interval</td>
<td>60</td>
</tr>
<tr>
<td>Sickness check script</td>
<td>autofs</td>
</tr>
<tr>
<td>Sickness check script timeout</td>
<td>10</td>
</tr>
<tr>
<td>Timeout</td>
<td>-1</td>
</tr>
</tbody>
</table>

[bright82->device*[bright82*]->services*[autofs*]]% set autostart yes
[bright82->device*[bright82*]->services*[autofs*]]% commit
[bright82->device[bright82]->services[autofs]]% category use default
[bright82->category[default]]% services
[bright82->category*[default*]->services*[autofs*]]% add autofs
[bright82->category*[default*]->services*[autofs*]]% set autostart yes
[bright82->category*[default*]->services*[autofs*]]% commit
[bright82->category*[default*]->services*[autofs*]]%
```

Filesystems imported to a regular node via an auto-mount operation must explicitly be excluded in `excludelistupdate` by the administrator, as explained in section 5.6.1, page 198.

The rest of this section describes the configuration of NFS for static mounts, using Bright View or cmsh.

Sections 3.10.1 and 3.10.2 explain how exporting and mounting of filesystems is done in general by an administrator using Bright View and cmsh, and considers some mounting behavior that the administrator should be aware of.

Section 3.10.3 discusses how filesystems in general on a diskless node can be replaced via mounts of NFS exports.

Section 3.10.4 discusses how the root (/) filesystem on a diskless node can be replaced via mounts of NFS exports.

Section 3.10.5 discusses how OFED InfiniBand or iWarp drivers can be used to provide NFS over RDMA.

### 3.10.1 Exporting A Filesystem Using Bright View And cmsh

**Exporting A Filesystem Using Bright View**

As an example, if an NFS volume exists at “bright82:/modeldata” it can be exported using Bright View as follows:

The `bright82` head node is selected via the clickpath `Devices→Head nodes[bright82]→Settings→Filesystem exports`. This shows the list of exports (figure 3.13):
Using the Add button, a new entry (figure 3.14) can be configured with values as shown:

- **Name**: Set arbitrarily to “Fluid Model Data”.
- **Path**: Set to `/modeldata`.
- **Network**: Set from the selection menu to allowing access to

![Image of NFS exports from a head node viewed using Bright View](image)

**Figure 3.13: NFS Exports From A Head Node Viewed Using Bright View**

![Image of configuring an NFS export using Bright View](image)

**Figure 3.14: Setting Up An NFS Export Using Bright View**

For this example, the value for “Name” is set arbitrarily to “Fluid Model Data”, the value for Path is set to `/modeldata`, and the value for Network is set from the selection menu to allowing access to
internalnet (by default 10.141.0.0/16 in CIDR notation).

By setting the Write option to Disabled, read-only access is kept.

Saving this preceding configuration means the NFS server now provides NFS access to this filesystem for internalnet.

The network can be set to other network values using CIDR notation. It can also be set to particular hosts such as just node001 and node002, by specifying a value of “node001 node002” instead. Other settings and options are also possible and are given in detail in the man pages for exports(5).

Exporting A Filesystem Using cmsh
The equivalent to the preceding Bright View NFS export procedure can be done in cmsh by using the fsexports submode on the head node (some output elided):

Example

```
[root@bright82 ~]# cmsh
[bright82]% device use bright82
[bright82->device[bright82]]% fsexports
[...->fsexports]% add "Fluid Model Data"
[...->fsexports[Fluid Model Data*]]% set path /modeldata
[...[Fluid Model Data*]]% set hosts 10.141.0.0/16
[...[Fluid Model Data*]]% commit
[...->fsexports[Fluid Model Data]]% list | grep Fluid
  Name (key)       Path         Hosts            Write
  ---------------- ------------- --------------- -----
  Fluid Model Data /modeldata 10.141.0.0/16 no
```

3.10.2 Mounting A Filesystem Using Bright View And cmsh
Continuing on with the export example from the preceding section, the administrator decides to mount the remote filesystem over the default category of nodes. Nodes can also mount the remote filesystem individually, but that is usually not a common requirement in a cluster. The administrator also decides not to re-use the exported name from the head node. That is, the remote mount name modeldata is not used locally, even though NFS allows this and many administrators prefer to do this. Instead, a local mount name of /modeldatagpu is used, perhaps because it avoids confusion about which filesystem is local to a person who is logged in, and perhaps to emphasize the volume is being mounted by nodes with GPUs.

Mounting A Filesystem Using Bright View
Thus, in Bright View, values for the remote mount point (bright82:/modeldata), the filesystem type (nfs), and the local mount point (/modeldatagpu) can be set in category mode, while the remaining options stay at their default values (figure 3.15):

© Bright Computing, Inc.
Saving the configuration saves the values, creating the local mount point, and the volume can now be accessed by nodes in that category.

**Mounting A Filesystem Using cmsh**

The equivalent to the preceding Bright View NFS mount procedure can be done in cmsh by using the fsmounts submode, for example on the default category. The add method under the fsmounts submode sets the mountpoint path, in this case /modeldatagpu (some output elided):

**Example**

[root@bright82 ~]# cmsh
[bright82]# category use default
[bright82->category[default]]% fsmounts
[bright82->category[default]->fsmounts]% add /modeldatagpu
[bright82->...*/modeldatagpu]% set device bright82:/modeldata
[bright82->...*/modeldatagpu]% set filesystem nfs
[bright82->category[default]->fsmounts[/modeldatagpu]]% commit
[bright82->category[default]->fsmounts[/modeldatagpu]]

<table>
<thead>
<tr>
<th>Device</th>
<th>Mountpoint (key)</th>
<th>Filesystem</th>
</tr>
</thead>
<tbody>
<tr>
<td>bright82:/modeldata</td>
<td>/modeldatagpu</td>
<td>nfs</td>
</tr>
</tbody>
</table>

[bright82->category[default]->fsmounts[/modeldatagpu]]% show

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Device</td>
<td>bright82:/modeldata</td>
</tr>
<tr>
<td>Dump</td>
<td>no</td>
</tr>
<tr>
<td>Filesystem</td>
<td>nfs</td>
</tr>
<tr>
<td>Filesystem Check</td>
<td>NONE</td>
</tr>
<tr>
<td>Mount options</td>
<td>defaults</td>
</tr>
<tr>
<td>Mountpoint</td>
<td>/modeldatagpu</td>
</tr>
</tbody>
</table>

Values can be set for “Mount options” other than default. For example, the “noac” flag can be added as follows:

[bright82->...*/modeldatagpu]% set mountoptions defaults,noac; commit
Mounting a CIFS might use:

```
[bright82->...[/modeldatagpu]]% set mountoptions gid,users,file_mode=0666,dir_mode=0777,
iocharset=iso8859-15,credentials=/path/to/credential
[bright82->...[/modeldatagpu*]]% commit
```

**General Considerations On Mounting A Filesystem**

There may be a requirement to segregate the access of nodes. For example, in the case of the preceding, because some nodes have no associated GPUs.

Besides the “**Allowed hosts**” options of NFS exports mentioned earlier in section 3.10.1, Bright Cluster Manager offers two more methods to fine tune node access to mount points:

- Nodes can be placed in another category that does not have the mount point.
- Nodes can have the mount point set, not by category, but per device within the `Nodes` resource. For this, the administrator must ensure that nodes that should have access have the mount point explicitly set.

Other considerations on mounting are that:

- **When adding a mount point object:**
  - The settings take effect right away by default on the nodes or node categories.
  - If `noauto` is set as a mount option, then the option only takes effect on explicitly mounting the filesystem.
  - If “`AutomaticMountAll=0`” is set as a CMDaemon directive (Appendix C), then CMDaemon changes for `/etc/fstab` are written to the file, but the `mount -a` command is not run by CMDaemon. However, the administrator should be aware that since `mount -a` is run by the distribution during booting, a node reboot implements the mount change.

- While a mount point object may have been removed, `umount` does not take place until reboot, to prevent mount changes outside of the cluster manager. If a `umount` needs to be done without a reboot, then it should be done manually, for example, using the `pdsh` or `pexec` command (section 13.1), to allow the administrator to take appropriate action if `umounting` goes wrong.

- **When manipulating mount points,** the administrator should be aware which mount points are inherited by category, and which are set for the individual node.
  - In Bright View, for a node, inheritance by category is indicated in the clickpath `Devices`→`Nodes`[**node name**]→`Edit`→`Settings`→`Filesystem mounts`, under the **INHERITED** column, with the entry (Category).
  - In `cmsh`, the category a mount belongs to is displayed in brackets. This is displayed from within the `fsmounts` submode of the `device` mode for a specified node:

**Example**

```
[root@bright82 ~]# cmsh -c "device; fsmounts node001; list"
```

<table>
<thead>
<tr>
<th>Device</th>
<th>Mountpoint (key)</th>
<th>Filesystem</th>
</tr>
</thead>
<tbody>
<tr>
<td>[default] none</td>
<td>/dev/pts</td>
<td>devpts</td>
</tr>
<tr>
<td>[default] none</td>
<td>/proc</td>
<td>proc</td>
</tr>
<tr>
<td>[default] none</td>
<td>/sys</td>
<td>sysfs</td>
</tr>
<tr>
<td>[default] none</td>
<td>/dev/shm</td>
<td>tmpfs</td>
</tr>
<tr>
<td>[default] $localnfserv+</td>
<td>/cm/shared</td>
<td>nfs</td>
</tr>
<tr>
<td>[default] bright82:/home</td>
<td>/home</td>
<td>nfs</td>
</tr>
<tr>
<td>bright82:/cm/shared/extra+</td>
<td>/home/examples</td>
<td>nfs</td>
</tr>
</tbody>
</table>

© Bright Computing, Inc.
To remove a mount point defined at category level for a node, it must be removed from within the category, and not from the specific node.

**Mount Order Considerations**

Care is sometimes needed in deciding the order in which mounts are carried out.

- For example, if both `/usr/share/doc` and a replacement directory subtree `/usr/share/doc/compat-gcc-34-3.4.6java` are to be used, then the stacking order should be that `/usr/share/doc` is mounted first. This order ensures that the replacement directory subtree overlays the first mount. If, instead, the replacement directory were the first mount, then it would be overlaid, inaccessible, and inactive.

- There may also be dependencies between the subtrees to consider, some of which may prevent the start up of applications and services until they are resolved. In some cases, resolution may be quite involved.

The order in which such mounts are mounted can be modified with the `up` and `down` commands within the `fsmounts` submode of `cmsh`.

### 3.10.3 Mounting A Filesystem Subtree For A Diskless Node Over NFS

#### NFS Vs `tmpfs` For Diskless Nodes

For diskless nodes (Appendix D.9), the software image (section 2.1.2) is typically installed from a provisioning node by the node-installer during the provisioning stage, and held as a filesystem in RAM on the diskless node with the `tmpfs` filesystem type.

It can be worthwhile to replace subtrees under the diskless node filesystem held in RAM with subtrees provided over NFS. This can be particularly worthwhile for less frequently accessed parts of the diskless node filesystem. This is because, although providing the files over NFS is much slower than accessing it from RAM, it has the benefit of freeing up RAM for tasks and jobs that run on diskless nodes, thereby increasing the cluster capacity.

An alternative “semi-diskless” way to free up RAM is to use a local disk on the node itself for supplying the subtrees. This is outlined in Appendix D.10.

#### Moving A Filesystem Subtree Out Of `tmpfs` To NFS

To carry out subtree provisioning over NFS, the subtrees are exported and mounted using the methods outlined in the previous examples in sections 3.10.1 and 3.10.2. For the diskless case, the exported filesystem subtree is thus a particular path under `/cm/images/<image>` on the provisioning node, and the subtree is mounted accordingly under `/` on the diskless node.

While there are no restrictions placed on the paths that may be mounted in Bright Cluster Manager 8.2, the administrator should be aware that mounting certain paths such as `/bin` is not possible.

When Bright View or `cmsh` are used to manage the NFS export and mount of the subtree filesystem, then `tmpfs` on the diskless node is reduced in size due to the administrator explicitly excluding the subtree from `tmpfs` during provisioning.

An example might be to export `/cm/images/default-image` from the head node, and mount the directory available under it, `/usr/share/doc`, at a mount point `/usr/share/doc` on the diskless node. In `cmsh`, such an export can be done by creating an FS export object corresponding to the software image object `defaultimage` with the following indicated properties (some prompt output elided):

**Example**

```
[root@bright82 ~]# cmsh
[bright82]% device use bright82; fsexports
[bright82->device[bright82]->fsexports]% add defaultimage
```

3 by default `<image>` is `default-image` on a newly-installed cluster

© Bright Computing, Inc.
### 3.10 Configuring NFS Volume Exports And Mounts

```bash
[bright82...defaultimage*]
% set path /cm/images/default-image
[bright82...defaultimage*]
% set hosts 10.141.0.0/16
[bright82...defaultimage*]
% commit
[bright82...defaultimage*]
% list | grep defaultimage
```

As the output to `list` shows, the NFS export should be kept read-only, which is the default. Appropriate parts of the export can then be mounted by a node or node category. The mount is defined by setting the mount point, the `nfs` filesystem property, and the export device. For example, for a node category (some output elided):

```bash
[bright82...defaultimage*]
% category use default
[bright82->category[default]]
% fsmounts
[bright82->...->fsmounts]
% add /usr/share/doc
[bright82->...->fsmounts]
% set device bright82:/cm/images/default-image/usr/share/doc
[bright82->...->fsmounts]
% set filesystem nfs
[bright82->...->fsmounts]
% commit
[bright82->category[default]->fsmounts/usr/share/doc]
% list
```

Other mount points can be also be added according to the judgment of the system administrator. Some consideration of mount order may be needed, as discussed on page 102 under the subheading “Mount Order Considerations”.

### An Example Of Several NFS Subtree Mounts

The following mounts save about 500MB from `tmpfs` on a diskless node with CentOS 6, as can be worked out from the following subtree sizes:

```bash
[root@bright82 ~] cd /cm/images/default-image/
[root@bright82 default-image]
```

<table>
<thead>
<tr>
<th>Subtree</th>
<th>Size</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>usr/share/locale</code></td>
<td>262M</td>
</tr>
<tr>
<td><code>usr/java</code></td>
<td>78M</td>
</tr>
<tr>
<td><code>usr/share/doc</code></td>
<td>107M</td>
</tr>
<tr>
<td><code>usr/src</code></td>
<td>45M</td>
</tr>
</tbody>
</table>

The filesystem mounts can then be created using the techniques in this section. After doing that, the result is then something like (some lines omitted):

```bash
[root@bright82 default-image]
% cmsh
[bright82%]
% category use default; fsmounts
[bright82->category[default]->fsmounts]
```

© Bright Computing, Inc.
Diskless nodes that have NFS subtree configuration carried out on them can be rebooted to start them up with the new configuration.

### 3.10.4 Mounting The Root Filesystem For A Diskless Node Over NFS

Mounting the root filesystem over NFS is a special case of mounting filesystem subtrees for a diskless node over NFS (section 3.10.3). The difference this time is that an initial root filesystem is deployed on the node via NFS as part of the standard Linux boot procedure. Some tmpfs mounts are then overlaid on top of parts of this filesystem.

The node being configured must have a disk setup that is diskless (section 3.9.4) for its node or node category.

If configuring a node category, it can be configured as follows on a CentOS 6 or 7 system:

1. The full diskless disk setup is partitioned first:
   ```
   [root@bright82 ~]# cmsh
   [bright82]# category use default
   [bright82->category[default]]% set disksetup /cm/images/default-image/cm/local/apps/cmd/etc/htdocs/disk-setup/slave-diskless.xml
   [bright82->category[default]]% commit #puts full OS in RAM
   ```

2. Then the default image is exported to the internal network:
   ```
   [bright82->category[default]]% device use master; fsexports
   [bright82->device[bright82]->fsexports]% add /cm/images/default-image
   [bright82->device[bright82]->fsexports]% set hosts internalnet
   [bright82->device[bright82]->fsexports]% set name default@internal
   [bright82->device[bright82]->fsexports[default@internal]]% commit #exports default image
   [bright82->device[bright82]->fsexports[default@internal]]% quit
   ```

3. The exported root filesystem is now mounted over the initial root filesystem:
   ```
   [root@bright82 ~]# cmsh -c "category use default; fsmounts; add /; set\device master:/cm/images/default-image; set mountoptions defaults,ro;\set filesystem nfs; commit" #adds readonly root via nfs
   ```

   The root filesystem should be read-only during normal use. However, when the main system starts up after provisioning, a distribution start up script mounts the root filesystem as read-write. One way to get it back to a read-only state again is to set a finalize script (section 3.15.4) that arranges for the drive to remount itself as read-only after the system is fully up. The remount can be done with an `rc.local` file, which runs after all the other `rc` start up scripts have run. A suitable finalize script is then:

   ```bash
   #!/bin/bash
   echo "mount -o remount,ro /" >> /localdisk/etc/rc.local
   ```

4. The filesystem for `/` however has plenty of parts that need to be writable too. These are now mounted back in minimal subtrees into RAM. Keeping these mounts minimal means that the RAM used by the node is minimal. These read/write subtrees can be specified in a file, with one line per subtree:
3.10 Configuring NFS Volume Exports And Mounts

[root@bright82 ~]# cat mountfiles  #for centos 6
/var/cache
/var/log
/var/lock
/var/run
/var/tmp
/var/spool
/tmp
/dev
/cm/local/apps/cmd/etc
/cm/local/apps/openldap/etc
/cm/local/apps/sge/var
/cm/local/apps/slurm/var
/cm/local/apps/pbspro/var
/cm/local/apps/pbspro-ce/var
/etc

For CentOS 7 the subtrees are slightly different:

[root@bright82 ~]# cat mountfiles  #for centos 7
/var/cache
/var/log
/var/tmp
/var/spool
/tmp
/dev
/cm/local/apps/cmd/etc
/cm/local/apps/openldap/etc
/cm/local/apps/sge/var
/cm/local/apps/slurm/var
/cm/local/apps/pbspro/var
/cm/local/apps/pbspro-ce/var
/etc
/run
/var/lib  #required by postfix and rpc-statd

The various workload manager /var directories for SGE/Slurm/PBS Pro/PBS Pro CE are only needed if these workload managers are to be set up and used.

The subtrees can be mounted with a for loop:

[root@bright82 ~]# (echo "category use default; fsmounts"
  for i in <$<mountfiles>
  do
    echo "add $i; set device tmpfs; set filesystem tmpfs; exit"
  done
  echo "commit" ) | cmsh

If there is further software in the root filesystem that needs write access, that too should be mounted back into RAM.

The diskless nodes cannot be powered off with a simple poweroff or rebooted with a reboot. This is because the parts of the filesystem required to carry out these actions are unmounted before they are called in the diskless configuration. The -f|--force option to these commands forces a poweroff or reboot, but should only be used after first cleanly unmounting any writable shared filesystems, such as /cm/shared and /home. This is because the forcing options interrupt I/O syncing when invoked, which can result in a corrupted filesystem.

© Bright Computing, Inc.
High Availability (Chapter 14) configured on the head nodes, with diskless nodes using a root filesystem mounted over NFS, would not work in a simple configuration such as this because it will result in stale NFS file handles over all the diskless nodes during a failover.

3.10.5 Configuring NFS Volume Exports And Mounts Over RDMA With OFED Drivers

If running NFS over RDMA, then at least NFS version 4.0 is recommended. NFS version 3 will also work with RDMA, but uses IPoIB encapsulation instead of native verbs. NFS version 4.1 uses the RDMA Connection Manager (librdmacm), instead of the InfiniBand Connection Manager (ib_cm) and is thereby also able to provide pNFS.

The administrator can set the version of NFS used by the cluster by setting the value of NFSVERS in the file /etc/nfsmount.conf on all the nodes, including the head node.

Drivers To Use For NFS over RDMA Must Be From The Parent Distribution

The use of the RDMA protocol (section 3.6) to provide NFS, by installing updated Bright Cluster Manager OFED drivers (section 7.6 of the Installation Manual) is currently not supported. This is because these drivers are packaged by Bright Computing from the vendor (Mellanox or Qlogic) releases, and the vendor releases themselves do not support NFS over RDMA.

However, NFS over RDMA does work within Bright Cluster Manager with the OFED drivers that the parent distribution provides, in the case of RHEL6/SL6/CENTOS6. For the remaining distributions, this option can be selected, but NFS will fall back to using the default NFS TCP/IP protocol.

When using NFS over RDMA, ibnet, the IP network used for InfiniBand, should be set. Section 3.6.3 explains how that can be done.

Exporting With Bright View And cmsh Using NFS Over RDMA

With the drivers installed, a volume export can be carried out using NFS over RDMA.

The procedure using Bright View is much the same as done in section 3.10.1 ("Exporting A Filesystem Using Bright View"), except for that the ibnet network should be selected instead of the default internalnet, and the "RDMA" option should be enabled.

The procedure using cmsh is much the same as done in section 3.10.1 ("Exporting A Filesystem Using cmsh"), except that the ibnet network (normally with a recommended value of 10.149.0.0/16) should be set, and the rdma option should be set.

Example

(based on the example in section 3.10.1)

```
...[...->fsexports*[Fluid Model Data*]]% set path /modeldata
[...[Fluid Model Data*]]% set hosts ibnet
[...[Fluid Model Data*]]% set rdma yes
[...[Fluid Model Data*]]% commit
...
```

Mounting With Bright View And cmsh Using NFS Over RDMA

The mounting of the exported filesystems using NFS over RDMA can then be done.

The procedure using Bright View is largely like that in section 3.10.2, ("Mounting A Filesystem Using Bright View"), except that the Device entry must point to master.ib.cluster so that it resolves to the correct NFS server address for RDMA, and the checkbox for NFS over RDMA must be ticked.

The procedure using cmsh is similar to that in section 3.10.2, ("Mounting A Filesystem Using cmsh"), except that device must be mounted to the ibnet, and the rdma option must be set, as shown:

Example

(based on the example in section 3.10.2)
3.11 Managing And Configuring Services

3.11.1 Why Use The Cluster Manager For Services?

The unix services can be managed from the command line using the standard distribution tools: chkconfig and

\[\text{service} \quad \text{<service name> } \text{start|stop|status|...} \]

where <service name> indicates a service such as mysqld, mariadb, nfs, postfix and so on.

Already installed services can also be brought under Bright Cluster Manager control, and started and stopped with Bright View and cmsh tools.

An additional convenience that comes with the cluster manager tools is that some CMDaemon parameters useful for managing services in a cluster are very easily configured, whether on the head node, a regular node, or for a node category. These parameters are:

- monitored: checks periodically if a service is running. Information is displayed and logged the first time it starts or the first time it dies

- autostart: restarts a failed service that is being monitored.
  - If autostart is set to on, and a service is stopped using Bright Cluster Manager, then no attempts are made to restart the service. Attempted autostarts become possible again only after Bright Cluster Manager starts the service again.
  - If autostart is set to on, and if a service is removed using Bright Cluster Manager, then the service is stopped before removal.
  - If autostart is off, then a service that has not been stopped by CMDaemon still undergoes an attempt to restart it, if
    * CMDaemon is restarted
    * its configuration files are updated by CMDaemon, for example in other modes, as in the example on page 66.

- runif: (only honored for nodes used as part of a high-availability configuration (chapter 14)) whether the service should run with a state of:
  - active: run on the active node only
  - passive: run on the passive only
  - always: run both on the active and passive
  - preferpassive: preferentially run on the passive if it is available. Valid only for head nodes. Invalid for failover groups (section 14.5.3).

The details of a service configuration remain part of the configuration methods of the service software itself.

- Thus Bright Cluster Manager can run actions on typical services only at the generic service level to which all the unix services conform. This means that CMDaemon can run actions such as
starting and stopping the service. If the restarting action is available in the script, then CM-
Daemon can also run that. An init script example can be found within the file /usr/share/doc/initscripts-<version>/sysvinitfiles in RHEL6 and derivatives.

- The operating system configuration of the service itself, including its persistence on reboot, re-
mains under control of the operating system, and is not handled by CMDaemon. So, stopping a
service within CMDaemon means that by default the service will still start up on reboot. Running
chkconfig from the command line can be used to configure the service to no longer start up on
reboot.

Bright Cluster Manager can be used to keep a service working across a failover event with an
appropriate runif setting and appropriate failover scripts such as the Prefailover script and the
Postfailover script (section 14.4.6). The details of how to do this will depend on the service.

3.11.2 Managing And Configuring Services—Examples

If, for example, the CUPS software is installed ("yum install cups"), then the CUPS service can be
managed in several ways:

Managing The Service From The Regular Shell, Outside Of CMDaemon

Standard unix commands from the bash prompt work, as shown by this session:

```
[root@bright82 ~]# chkconfig cups on
[root@bright82 ~]# service cups start
```

Managing The Service From cmsh

Starting the service in cmsh: The following session illustrates adding the CUPS service from within
device mode and the services submode. The device in this case is a regular node, node001, but a head
node can also be chosen. Monitoring and auto-starting are also set in the session (some lines elided):

```
[bright82]% device services node001
[bright82->device[node001]->services]% add cups
[bright82->device*[node001*]->services*[cups*]]% show
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Autostart</td>
<td>no</td>
</tr>
<tr>
<td>Belongs to role</td>
<td>no</td>
</tr>
<tr>
<td>Monitored</td>
<td>no</td>
</tr>
<tr>
<td>Run if</td>
<td>ALWAYS</td>
</tr>
<tr>
<td>Service</td>
<td>cups</td>
</tr>
</tbody>
</table>

[bright82->device*[node001*]->services*[cups*]]% set monitored on
[bright82->device*[node001*]->services*[cups*]]% set autostart on
[bright82->device*[node001*]->services*[cups*]]% commit
[bright82->device[node001]->services[cups]]%

Apr 14 14:02:16 2017 [notice] node001: Service cups was started
[bright82->device[node001]->services[cups]]%
```

Other service options in cmsh: Within cmsh, the start, stop, restart, and reload options to the
service <service name> start|stop|restart|...
command can be used to manage the service at the services submode level. For example, continu-
ing with the preceding session, stopping the CUPS service can be done by running the cups service
command with the stop option as follows:

```
[bright82->device[node001]->services[cups]]% stop
```

© Bright Computing, Inc.
The service is then in a STOPPED state according to the status command.

```
[bright82->device[node001]->services[cups]]% status
  cups [STOPPED]
```

Details on how a state is used when monitoring a service are given in the section “Monitoring A Service With cmsh And Bright View” on page 110.

Continuing from the preceding session, the CUPS service can also be added for a node category from category mode:

```
[bright82->device[node001]->services[cups]]% category
bright82->category]% services default
[bright82->category[default]->services]% add cups
```

As before, after adding the service, the monitoring and autostart parameters can be set for the service. Also as before, the options to the service <service name> start|stop|restart|... command can be used to manage the service at the services submode level. The settings apply to the entire node category (some lines elided):

```
Example

[bright82->category*[default*]->services*[cups*]]% show
...
[bright82->category*[default*]->services*[cups*]]% set autostart yes
[bright82->category*[default*]->services*[cups*]]% set monitored yes
[bright82->category*[default*]->services*[cups*]]% commit
[bright82->category*[default*]->services*[cups*]]% status
  node001................. cups [STOPPED ]
  node002................. cups [ UP ]
  node003................. cups [ UP ]
  node004................. cups [ UP ]
  node005................. cups [ UP ]
```

Managing The Service From Bright View

Using Bright View, a service can be managed from a Services option, accessible from the Settings option of

- Head Nodes, for example via a clickpath of Devices→Head Nodes[bright82]→Settings→Services
- Nodes, for example via a clickpath of Devices→Nodes[node001]→Settings→Services
- Node categories, for example via a clickpath of Grouping→Node categories[default]→Settings→Services

© Bright Computing, Inc.
Figure 3.16 shows the Services subwindow accessed from the default software image, which is an item within the “Node Categories” subwindow. By default, there are no services set by the category for nodes.

The service `<service name>` start|stop|restart... command options start, stop, restart, and so on, are displayed as options in the service configuration window.

The existence of the service itself can be managed using the Add, Remove, and, if the service already exists, the Edit button. The change can be reverted with the Revert button.

Figure 3.17 shows CUPS being set up from an Add dialog in the services subwindow. The subwindow is accessible via the [Add] button of figure 3.16.

For a service in the services subwindow, clicking on the Status button in figure 3.16 displays a grid of the state of services on a running node as either Up or Down.

**Monitoring A Service With cmsh And Bright View**

The service is in a DOWN state if it is not running, and in a FAILING state if it is unable to run after 10 auto-starts in a row. Event messages are sent during these first 10 auto-starts. After the first 10 auto-starts, no more event messages are sent, but autostart attempts continue.

In case an autostart attempt has not yet restarted the service, the reset option may be used to attempt
3.12 Managing And Configuring A Rack

an immediate restart. The reset option is not a service option in the regular shell, but is used by CMDaemon (within cmsh and Bright View) to clear a FAILING state of a service, reset the attempted auto-starts count to zero, and attempt a restart of the service.

The monitoring system sets the ManagedServicesOk health check (Appendix G.2.1) to a state of FAIL if any of the services it monitors is in the FAILING state. In cmsh, the statuses of the services are listed by running the latesthealthdata command (section 12.6.3) from device mode.

Standard init.d script behavior is that the script return a non-zero exit code if the service is down, and a zero exit code if the service is up. A non-zero exit code makes Bright Cluster Manager decide that the service is down, and should be restarted.

However, some scripts return a non-zero exit code even if the service is up. These services therefore have Bright Cluster Manager attempt to start them repetitively, even though they are actually running.

This behavior is normally best fixed by setting a zero exit code for when the service is up, and a non-zero exit code for when the service is down.

Removing A Service From CMDaemon Control Without Shutting It Down

Removing a service from Bright Cluster Manager control while autostart is set to on stops the service on the nodes:

```
[bright82->category[default]->services]% add cups
[bright82->category*[default*]->services*[cups*]]% set monitored on
[bright82->category*[default*]->services*[cups*]]% set autostart on
[bright82->category*[default*]->services*[cups*]]% commit; exit
Wed May 23 12:53:58 2012 [notice] node001: Service cups was stopped
```

In the preceding cmsh session, cups starts up when the autostart parameter is committed, if cups is not already up.

The behavior of having the service stop on removal is implemented because it is usually what is wanted.

However, sometimes the administrator would like to remove the service from CMDaemon control without it shutting down. To do this, autostart must be set to off first.

```
[bright82->category[default]->services]% add cups
[bright82->category*[default*]->services*[cups*]]% set monitored on
[bright82->category*[default*]->services*[cups*]]% set autostart off
[bright82->category*[default*]->services*[cups*]]% commit; exit
Wed May 23 12:54:40 2012 [notice] node001: Service cups was started
[bright82->category*[default*]->services][cups*]]% remove cups; commit
[bright82->category*[default*]->services]% !# no change: cups stays up
```

3.12 Managing And Configuring A Rack

3.12.1 Racks

A cluster may have local nodes grouped physically into racks. A rack is 42 units in height by default, and nodes normally take up one unit.

Racks Overview

Racks overview in Bright View: The Racks overview pane can be opened up in Bright View via the clickpath Datacenter Infrastructure→Racks (figure 3.18). Racks can then be added, removed, or edited from that pane.

Within the Racks pane:

- a new rack item can be added with the Add button.

© Bright Computing, Inc.
• an existing rack item can be edited with the Edit button, or by double-clicking on the item itself in the pane.

These actions bring up the rack configuration pane (figure 3.19).

**Racks overview in cmsh:** The rack mode in cmsh allows racks defined in the cluster manager to be listed:

```plaintext
[bright82->rack] % list
Name (key) Room x-Coordinate y-Coordinate Height
-------------- ------------- ------------- ------------- ------
rack2 skonk works 2 0 42
racknroll 1 0 42
```

**Rack Configuration Settings**

**Rack configuration settings in Bright View:** The Settings pane for editing a rack item selected from the Racks pane is shown in figure 3.19.

The rack item configuration settings are:

- **Name:** A unique name for the rack item. Names such as rack001, rack002 are a sensible choice.
• **Room**: A unique name for the room the rack is in.

• **Position**: The $x$- and $y$-coordinates of the rack in a room. These coordinates are meant to be a hint for the administrator about the positioning of the racks in the room, and as such are optional, and can be arbitrary. The **Notes** field can be used as a supplement or as an alternative for hints.

• **Height**: by default this is the standard rack size of 42U.

• **Bottom of rack is position 1**: Normally, a rack uses the number 1 to mark the top and 42 to mark the bottom position for the places that a device can be positioned in a rack. However, some manufacturers use 1 to mark the bottom instead. Ticking the checkbox records the numbering layout accordingly for all racks, if the checkboxed rack is the first rack seen in **Rackview**.

**Rack configuration settings in cmsh**: In cmsh, tab-completion suggestions for the **set** command in **rack** mode display the racks available for configuration. On selecting a particular rack (for example, **rack2** as in the following example), tab-completion suggestions then display the configuration settings available for that rack:

Example

```
[bright82->rack]% set rack
rack1 rack2 rack3
[bright82->rack]% set rack2
height name revision width y-coordinate inverted notes room x-coordinate
```

The configuration settings for a particular rack obviously match with the parameters associated with and discussed in figure 3.19. The only slightly unobvious match is the Boolean parameter **inverted** in cmsh, which simply corresponds directly to “**Bottom of rack is position 1**” in Bright View.

Setting the values can be done as in this example:

Example

```
[bright82->rack]% use rack2
[bright82->rack][rack2]% set room "skonk works"
[bright82->rack][rack2]% set x-coordinate 2
[bright82->rack][rack2]% set y-coordinate 0
[bright82->rack][rack2]% set inverted no
[bright82->rack][rack2]% commit
[bright82->rack][rack2]%
```

3.12.2 **Assigning Devices To A Rack**

Devices such as nodes, switches, and chassis, can be assigned to racks.

By default, no such devices are assigned to a rack.

Devices can be assigned to a particular rack and to a particular position within the rack as follows:

**Assigning Devices To A Rack Using Bright View**

Using Bright View, a device such as a node **node001** can be assigned to a rack via the clickpath **Devices → Nodes [node001] → Settings → Rack** (figure 3.20):
Assigning Devices To A Rack Using cmsh

Using cmsh, the assignment can be done to a rack as follows:

```
[bright82->device]% foreach -n node001..node003 (set deviceheight 1; set
deviceposition 2; set rack rack2)
[bright82->device*]% commit
Successfully committed 3 Devices
[bright82->device]%
```

The Convention Of The Top Of The Device Being Its Position

Since rack manufacturers usually number their racks from top to bottom, the position of a device in a rack (using the parameter `Position` in Bright View, and the parameter `deviceposition` in cmsh) is always taken to be where the top of the device is located. This is the convention followed even for the less usual case where the rack numbering is from bottom to top.

A position on a rack is 1U of space. Most devices have a height that fits in that 1U, so that the top of the device is located at the same position as the bottom of the device, and no confusion is possible. The administrator should however be aware that for any device that is greater than 1U in height such as, for example, a blade enclosure chassis (section 3.12.3), the convention means that it is the position of the top of the device that is where the device is considered to be. The position of the bottom of the device is ignored.

### 3.12.3 Assigning Devices To A Chassis

**A Chassis As A Physical Part Of A Cluster**

In a cluster, several local nodes may be grouped together physically into a chassis. This is common for clusters using blade systems. Clusters made up of blade systems use less space, less hardware, and less electrical power than non-blade clusters with the same computing power. In blade systems, the blades are the nodes, and the chassis is the blade enclosure.

A blade enclosure chassis is typically 6 to 10U in size, and the node density for server blades is typically 2 blades per unit with 2014 technology.
Chassis Configuration And Node Assignment

Basic chassis configuration and node assignment with Bright View: Adding and removing chassis from a list of chassis in Bright View can be done via the clickpath Devices → Chassis. This opens up a Chassis pane that provides an overview of the list of chassis.

Within the Chassis pane, each chassis can further have an individual subpane opened for it. Each individual chassis configuration can be added, edited, or deleted via this chassis subpane (figure 3.21).

![Figure 3.21: Bright View Chassis Subpane](image)

The options that can be set within the subpane include the following:

- **Hostname**: a name that can be assigned to the chassis operating system
- **Mac**: the MAC address of the chassis
- **Rack**: the rack in which the chassis is placed
- **Members**: the Members menu option allows devices to be assigned to a chassis (figure 3.22). An item within the Device set can be any item from the subsets of Node, Switch, Power Distribution Unit, Generic Device, Rack Sensor, and GPU Unit. These items can be filtered for viewing, depending on whether they are Assigned (members of the chassis), Not assigned (not members of the chassis), or they can All be viewed (both Assigned and Not assigned items).
- **Layout**: how the nodes in a chassis are laid out visually.
- **Network**: which network the chassis is attached to.
- **Username, Password**: the user name and password to access the chassis operating system
- **Power control, Custom power script, Custom power script argument**: power-related items for the chassis.
- **Userdefined1, Userdefined2**: administrator-defined variables that can be used by CMDaemon.
Basic chassis configuration and node assignment with cmsh: The chassis mode in cmsh allows configuration related to a particular chassis. Tab-completion suggestions for a selected chassis with the set command show possible parameters that may be set:

Example

[bright82->device[chassis1]]% set
containerindex    ip    powerdistributionunits
custompingscript  layout  rack
custompingscriptargument  mac  revision
custompowerscript  members  slots
custompowerscriptargument  model  tag
defaultgateway  network  userdefined1
deviceheight  notes  userdefined2
deviceposition  partition  username
ethernetswitch  password
hostname  powercontrol

Whether the suggested parameters are actually supported depends on the chassis hardware. For example, if the chassis has no network interface of its own, then the ip and mac address settings may be set, but cannot function.

The “positioning” parameters of the chassis within the rack can be set as follows with cmsh:

Example

[bright82->device[chassis1]]% set rack rack2
[bright82->device*[chassis1*]]% set deviceposition 1; set deviceheight 6
[bright82->device*[chassis1*]]% commit

The members of the chassis can be set as follows with cmsh:

Example

[bright82->device[chassis1]]% append members bright82 node001..node005
[bright82->device*[chassis1*]]% commit
3.13 Configuring A GPU Unit, And Configuring GPU Settings

3.13.1 GPUs And GPU Units

GPUs (Graphics Processing Units) are processors that are heavily optimized for executing certain types of parallel processing tasks. GPUs were originally used for rendering graphics, and one GPU typically has hundreds of cores. When used for general processing, they are sometimes called General Processing GPUs, or GPGPUs. For convenience, the “GP” prefix for General Processing is not used in this manual.

A GPU is typically placed on a PCIe card. GPUs can be physically inside the node that uses them, or they can be physically external to the node that uses them. As far as the operating system on the node making use of the physically external GPUs is concerned, the GPUs are internal to the node.

If the GPUs are physically external to the node, then they are typically in a GPU unit. A GPU unit is a chassis that hosts only GPUs. It is typically able to provide GPU access to several nodes, usually via PCIe extender connections. This ability means that external GPUs typically require more configuration than internal GPUs.

3.13.2 GPU Unit Configuration Example: The Dell PowerEdge C410x

An example of a GPU unit is the Dell PowerEdge C410x, which comes in a 3U chassis size, has up to 16 Tesla M-series GPUs (in the form of cards in slots) inside it, and can allocate up to 4 GPUs per node.

It can be configured to work with Bright Cluster Manager as follows:

1. The GPUs in the unit are assigned to nodes using the direct web interface provided by the Baseboard Management Controller (BMC) of the C410x. This configuration is done outside of Bright Cluster Manager. The assignment of the GPUs can be done only according to the fixed groupings permitted by the web interface.

For example, if the unit has 16 GPUs in total (1, 2, ..., 16), and there are 4 nodes in the cluster (node001, node002, ..., node004), then an appropriate GPU indices mapping to the node may be:

- node001 is assigned 1, 2, 15, 16
- node002 is assigned 3, 4, 13, 14
- node003 is assigned 5, 6, 11, 12
- node004 is assigned 7, 8, 9, 10

This mapping is decided by accessing the C410x BMC with a browser, and making choices within the “Port Map” resource (figure 3.23). 4 mappings are displayed (Mapping 1, 2, 3, 4), with columns displaying the choices possible for each mapping. In the available mapping choices, the iPass value indicates a port, which corresponds to a node. Thus iPass 1 is linked to node001, and its corresponding PCIE values (1, 2, 15, 16) are the GPU indices here. Each iPass port can have 0 (N/A), 2, or 4 PCIE values associated with it, so that the GPU unit can supply up to 4 GPUs to each node.

The GPU indices (PCIE values) (that is, the numbers 1, 2, ..., 16) are used to identify the card in Bright Cluster Manager. The lowest GPU index associated with a particular node—for example, 5 (rather than 6) for node03—is used to make Bright Cluster Manager aware in the next step that an association exists for that node, as well as aware which the first GPU card associated with it is.

2. The GPUs can be assigned to the nodes in Bright Cluster Manager as follows:

First, the GPU unit holding the GPUs can be assigned to the GPU unit resource. This can be carried out by adding the hostname of the GPU unit, for example in cmsh with:

```
[root@bright82 ~]# cmsh
[bright82]% device
[bright82->device]% add gpuunit schwartz
```
This drops the administrator into the gpuunit object, which has the hostname schwartz here. The equivalent addition in Bright View can be done via Devices→GPU Units→Add button.

Next, the IP address, network, and BMC user name and password can be set to appropriate values:

**Example**

```
[bright82->device[ schwartz ]]% set ip 10.148.0.10
[bright82->device[ schwartz ]]% set network bmcnet
[bright82->device[ schwartz ]]% bmcsettings
[bright82->device[ schwartz ]->bmcsettings]% set username darkhelmet
[bright82->device[ schwartz ]->bmcsettings]% set userid 1002
[bright82->device[ schwartz ]->bmcsettings]% set password 12345
[bright82->device[ schwartz ]->bmcsettings]% commit
```

Here, it is assumed that a BMC network that the BMC network interface is connected to has already been created appropriately. If it is not already set up, then the network can be configured as described in section 3.7.

The value of owners must be set to the list of nodes that have access to GPUs.

Also, the reserved variable, userdefined1 must be set to a key=value pair, so that each key (the node in owner) has a value (the PCIE values), assigned to it:

**Example**

```
[bright82->device[ schwartz ]]% set owners node001 node002 node003 node004
[bright82->device[ schwartz ]]% set userdefined1 node001=1,2,15,16 node002=3,4,13,\ 14 node003=5,6,11,12 node004=7,8,9,10
[bright82->device[ schwartz ]]% commit
```
3.13 Configuring A GPU Unit, And Configuring GPU Settings

As a convenience, if the indices are all separated by 1, as in the case of node004 here, only the lowest index need be set for that node. The line setting the userdefined1 value can thus equivalently be carried out with:

**Example**

```
[bright82->device[ schwartz*]]% set userdefined1 node001=1,2,15,16 node002=3,4,13,14 node003=5,6,11,12 node004=7
```

Once these values are committed, Bright Cluster Manager is able to track GPU assignment consistently and automatically.

3. Finally, CUDA drivers should be installed on the relevant nodes so that they can make use of the GPUs. The details on how to do this are given in section 7.4 of the *Installation Manual*.

### 3.13.3 Configuring GPU Settings

**The gpusettings Submode In cmsh**

In cmsh, GPUs can be configured for a specified node via device mode.

Going into the gpusettings submode for that node then allows a type of GPU to be set, from the amd or nvidia types, and a range to be specified for the GPU slots:

**Example**

```
[root@bright82 ~]# cmsh
[bright82]% device use node001
[bright82->device[node001]]% gpusettings
[bright82->device[node001]->gpusettings]% add nvidia 1-3 ; commit
```

The range can be specified as

- a single number, for example: 3
- a range, for example: 0-2
- all, using:
  - all
  - or
  - *

GPUs can also be configured for a specified category via category mode. For example, using the category default, then entering into the gpusettings submode allows a type (nvidia or amd) and a range to be set for the range of GPUs:

**Example**

```
[root@bright82 ~]# cmsh
[bright82]% category use default
[bright82->category[default]]% gpusettings
[bright82->category[default]->gpusettings]% list
GPU range (key) Power limit ECC mode Compute mode Clock speeds
----------------- ------------ ------------------------- ------------
[bright82->category[default]->gpusettings]% add nvidia 1-3 ; commit
[bright82->category[default]->gpusettings[nvidia:1-3]]% show
Parameter                Value
-------------------------------
```

© Bright Computing, Inc.
As usual, GPU settings for a node override those for a category (section 2.1.3).

**GPU Settings With NVIDIA GPUs**

The installation of the NVIDIA GPU software driver is covered in section 7.4.1 of the *Installation Manual*. The present section is about configuring NVIDIA GPUs in Bright Cluster Manager. The driver itself does not necessarily have to be in place for the configuration to be done, although the configuration only becomes active when the driver is installed.

After a GPU type has been set, the following NVIDIA GPU settings may be specified, if supported, from within the `gpusettings` submode:

- **clockspeeds**: The pair of clock speeds (frequency in MHz) to be set for this parameter can be selected from the list of available speeds. The available speeds can be seen by running the `status` command. The values are specified in the form: `<number for GPU processor>,<number for memory>`

- **clocksyncboostmode**: GPU boosting. Exceed the maximum core and memory clock speeds if it is safe. Choices are:
  - `enabled`
  - `disabled`

- **computemode**: Contexts can be computed with the following values for this mode:
  - **Default**: Multiple contexts are allowed
  - **Exclusive thread**: Only one context is allowed per device, usable from one thread at a time
  - **Exclusive process**: Only one context is allowed per device, usable from multiple threads at a time. This mode option is valid for CUDA 4.0 and higher. Earlier CUDA versions ran only in this mode.
  - **Prohibited**: No contexts are allowed per device

- **eccmode**: Sets the ECC bit error check, with:
  - `enabled`
  - `disabled`

  When ECC is enabled:
  - Single bit errors are detected, using the `EccSBitGPU` metric (page 699), and corrected automatically.
  - Double bit errors are also detected, using the `EccDBitGPU` metric (page 699), but cannot be corrected.

The `gpureset` and `gpuclearecc` commands (page 121) are relevant for this setting.

- **gpureset**: After changing the `eccmode` setting, the GPU must be reset to make the new value active.
- **gpuclearecc**: This is used to clear ECC counts for the GPU.

- **GPU range**: range values can be set as follows:
3.13 Configuring A GPU Unit, And Configuring GPU Settings

- all: The GPU settings apply to all GPUs on the node.
- <number>: The GPU settings apply to an individual GPU, for example: 1
- <number range>: The GPU settings apply to a range of GPUs, for example: 1,3-5

- **powerlimit**: The administrator-defined upper power limit for the GPU. Only valid if `powermode` is supported.
  - `min`: The minimum upper power limit that the hardware supports.
  - `max`: The maximum upper power limit that the hardware supports.
  - `<number>`: An arbitrary upper power limit, specified as a number between `min` and `max`
  - `default`: Upper power limit based on the default hardware value.

If no value is specified for a GPU setting, then the hardware default is used.

The `gpureset` and `gpuclearecc` commands are commands for NVIDIA GPUs that can be run within `cmsh` at a node level, at category level, and also for other groupings of nodes. The `cmsh` help text for these commands (`help gpureset` and `help gpuclearecc`) gives details, including node range specifications.

- The `gpureset` command carries out a power reset. It is executed only if there are no processes using the GPU. Possible processes using the GPU can be CUDA applications, X, monitoring applications, or other calculations. The device is all done with the reset in about 2 seconds.

  **Example**

  ```bash
  [bright82->device]% gpureset -c default 1,3-6
  ```

  This resets GPUs 1, 3, 4, 5, and 6 in the default category.

  The command can be used to reset GPUs without rebooting the node, or to reset some hung GPUs.

- The `gpuclearecc` command clears the ECC counts for the GPU. The ECC counts are of two types:
  - `volatile`: Resets when the driver unloads. Resets on power cycle.
  - `aggregate`: Persists when the driver unloads. Resets on power cycle.

  By default, both types are cleared when the command is run, but each type can be cleared separately.

  **Example**

  ```bash
  [bright82->device]% gpuclearecc -c default 1,3-6 -a
  ```

  This clears the aggregate ECC counts for the GPUs 1, 3, 4, 5, and 6 in the default category.

**GPU Settings With AMD GPUs**

GPU settings for AMD Radeon GPUs are accessed via `cmsh` in the same way as NVIDIA GPU settings. The AMD GPU setting parameters do differ from the NVIDIA ones.

The AMD GPUs supported are Radeon cards. A list of cards and operating systems compatible with the Linux driver used is at https://support.amd.com/en-us/kb-articles/Pages/Radeon-Software-for-Linux-Release-Notes.aspx

AMD GPU driver installation is described in section 7.5 of the *Installation Manual*.

The Radeon Instinct MI25 shows the following settings in Ubuntu 16.06 running a Linux 4.4.0-72-generic kernel:

© Bright Computing, Inc.
Example

[bright82->device[node001]->gpusettings]% list
Type GPU range Info
---- --------- ------------------
AMD 0 PowerPlay: manual
[bright82->device[node001]->gpusettings]% use amd:0
[bright82->device[node001]->gpusettings[amd:0]]% show
Parameter Value
-------------------------------- -----------------------------
Activity threshold 1
Fan speed 255
GPU clock level 5
GPU range 0
Hysteresis down 0
Hysteresis up 0
Info PowerPlay: manual
Memory clock level 3
Minimum GPU clock 0
Minimum memory clock 0
Overdrive percentage 1
PowerPlay mode manual
Revision
Type AMD

The possible values here are:

- **activitythreshold**: Percent GPU usage at a clock level that is required before clock levels change. From 0 to 100.

- **fanspeed**: Maximum fan speed. From 0 to 255

- **gpuclocklevel**: GPU clock level setting. From 0 to 7.

- **gprange**: The slots used.

- **hysteresisdown**: Delay in milliseconds before a clock level decrease is carried out.

- **hysteresisup**: Delay in milliseconds before a clock level increase is carried out.

- **info**: A compact informative line about the GPU status.

- **memoryclocklevel**: Memory clock speed setting. From 0-3. Other cards can show other values.

- **minimumgpuclock**: Minimum clock frequency for GPU, in MHz. The kernel only allows certain values. Supported values can be seen using the status command.

- **minimummemoryclock**: Minimum clock frequency for the memory, in MHz. The kernel only allows certain values. Supported values can be seen using the status command.

- **overdrivepercentage**: Percent overclocking. From 0 to 20%

- **powerplaymode**: Decides how the performance level power setting should be implemented.
  - **high**: keep performance high, regardless of GPU workload
  - **low**: keep performance low, regardless of GPU workload
  - **auto**: Switch clock rates according to GPU workload
  - **manual**: Use the memory clock level and GPU clock values.
The status command displays supported clock frequencies (some values ellipsized):

**Example**

```
[bright82->device[node001]->gpusettings[amd:0]]% status
Index Name          Property Value          Supported
----- ----------------- ------------ -------------------------------
0     Radeon Instinct MI25 Clock          1399Mhz 852Mhz, 991Mhz, ..., 1440Mhz, 1515Mhz
0     Radeon Instinct MI25 Memory        945Mhz  167Mhz, 500Mhz, 800Mhz, 945Mhz
```

**The gpusettings Submode In Bright View**

In Bright View the GPU settings can be accessed within the settings options for a device with a GPU. For example, if a regular node, node001, has a GPU, then the clickpath would be:

*Devices*→*Nodes*[node001]→*Edit*→*Settings*→*GPU Settings*

which opens up the GPU Settings subpane for that node (figure 3.24). Similarly, GPU settings can also be accessed within the *Category* resource, selecting a category item, and then selecting the *GPU Settings* subpane.

![Figure 3.24: GPU Settings Subpane For A Node](image)

**GPU Configuration For HPC Workload Managers**

**Slurm:** To configure GPUs for Slurm, changes are made in `slurm.conf`. When adding new parameters, care must be taken if these parameters are already in the file. In that case, the old parameters should be commented out or removed, because `slurmd` is unlikely to work properly with duplicated parameters.

The workload manager configuration requires the following steps:

1. Enabling and configuring the consumable resource allocation plugin in `slurm.conf`:

   **Example**
2. Enabling the GPU accounting tracker and enabling accounting via cgroup settings in `slurm.conf`.

   **Example**

   ```
   AccountingStorageTRES=gres/gpu
   JobAcctGatherType=jobacct_gather/cgroup
   ```

   These are not mandatory settings.

3. Restarting `slurmctld`

4. Going to the SlurmClient role, and adding generic resources, for each GPU:

   **Example**

   ```
   [bright82->category[dgx]->roles[slurmclient]->genericresources]% add gpu0
   [bright82->category[dgx+]->roles[slurmclient+]->genericresources*[gpu0*]]% set name gpu
   [bright82->category[dgx+]->roles[slurmclient+]->genericresources*[gpu0*]]% set file /dev/nvidia0
   [bright82->category[dgx+]->roles[slurmclient+]->genericresources*[gpu0*]]% commit
   ```

   (Repeat similar settings for the other GPUs, `gpu1...gpu7`)

   ```
   [bright82->category[dgx]->roles[slurmclient]->genericresources]% list
   ```

<table>
<thead>
<tr>
<th>Alias (key)</th>
<th>Name</th>
<th>Type</th>
<th>Count</th>
<th>File</th>
</tr>
</thead>
<tbody>
<tr>
<td>gpu0</td>
<td>gpu</td>
<td>/dev/nvidia0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>gpu1</td>
<td>gpu</td>
<td>/dev/nvidia1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>gpu2</td>
<td>gpu</td>
<td>/dev/nvidia2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>gpu3</td>
<td>gpu</td>
<td>/dev/nvidia3</td>
<td></td>
<td></td>
</tr>
<tr>
<td>gpu4</td>
<td>gpu</td>
<td>/dev/nvidia4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>gpu5</td>
<td>gpu</td>
<td>/dev/nvidia5</td>
<td></td>
<td></td>
</tr>
<tr>
<td>gpu6</td>
<td>gpu</td>
<td>/dev/nvidia6</td>
<td></td>
<td></td>
</tr>
<tr>
<td>gpu7</td>
<td>gpu</td>
<td>/dev/nvidia7</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

5. In the SlurmServer role, enabling the option to constrain devices:

   **Example**

   ```
   [bright82->device*[bright82*]->roles*[slurmservlet*]->cgroups*] set constraindevices yes
   [bright82->device*[bright82*]->roles*[slurmservlet*]->cgroups*] commit
   ```

6. By default, Slurm just allows a single job to be executed per node. To change this behavior, it is necessary to allow oversubscription. For example, to allow 8 jobs per node:

   **Example**

   ```
   [bright82]% jobqueue
   [bright82->jobqueue(slurm)] use defq
   [bright82->jobqueue(slurm)->defq] set oversubscribe yes:8
   [bright82->jobqueue*(slurm)->defq*] commit
   ```

© Bright Computing, Inc.
3.13 Configuring A GPU Unit, And Configuring GPU Settings

PBS Pro: Bright Cluster Manager does not support GPU configuration in PBS Pro for now.

UGE: GPU configuration is carried out as follows:

1. **UGE cgroups parameter modifications for GPU devices**
   In Bright Cluster Manager 8.0, 8.1, and 8.2: The command `qconf -mconf <hostname>` is used to add GPU device(s) to the cgroups_params parameter for the host.

   For example, if node001 has access to 8 GPUs, then running the command `qconf -mconf node001` on the head node may initially show a configuration such as:

   ```
   hostname node001
cgroups_params cgroup_path=/sys/fs/cgroup cpuset=true \mount=false freezer=true freeze_pe_tasks=true \killing=false forced_numa=false \h_vmem_limit=false m_mem_free_hard=false \m_mem_free_soft=false min_memory_limit=0
   ```

   The GPU devices can then be appended to the end of the last line and saved, so that the last line becomes:

   ```
   Example
   m_mem_free_soft=false min_memory_limit=0 devices=/dev/nvidia[0-7]
   ```

   The parameter can be made persistent by setting the AdvancedConfig directive (page 649) to include the setting `UGEDisableCgroupSettings=1` and restarting CMDaemon.

2. **UGE port configuration for DCGM**
   The DCGM port should be set correctly.

   It can be changed from its default with the command `qconf -mconf`. The format for the setting is:

   ```
   Example
   execd_params UGE_DCGM_PORT=<port>
   ```

   The default value for `<port>` is 5555.

LSF: allows GPU devices to be autodetected. To allow it, the administrator should set the following parameter in `lsf.conf`:

```
Example
LSF_GPU_AUTOCONFIG=Y
```

GPU resource enforcement can be configured for LSF as follows:

```
Example
[bright82->device[bright82]->roles[lsfserver]->cgroups] append resourceenforce gpu
[bright82->device[bright82*]->roles[lsfserver*]->cgroups*] commit
[bright82->device[bright82*]->roles[lsfserver*]->cgroups*]
```

© Bright Computing, Inc.
3.14 Configuring Custom Scripts

Some scripts are used for custom purposes. These are used as replacements for certain default scripts, for example, in the case of non-standard hardware where the default script does not do what is expected. The custom scripts that can be set, along with their associated arguments are:

- `custompowerscript` and `custompowerscriptargument`
- `custompingscript` and `custompingscriptargument`
- `customremoteconsolescript` and `customremoteconsolescriptargument`

The environment variables of CMDaemon (section 3.3.1 of the Developer Manual) can be used in the scripts. Successful scripts, as is the norm, return 0 on exit.

3.14.1 custompowerscript

The use of custom power scripts is described in section 4.1.4.

3.14.2 custompingscript

The following example script:

Example

```bash
#!/bin/bash
/bin/ping -c1 $CMD_IP
```

...can be defined and set for the cases where the default built-in ping script, cannot be used.

By default, the node device states are detected by the built-in ping script (section 5.5) using ICMP ping. This results in the statuses that can be seen on running the list command of cmsh in device mode. An example output, formatted for convenience, is:

Example

```
[root@bright82]# cmsh -c "device; format hostname:15, status:15; list"
hostname (key) status
--------------- --------------
bright82 [ UP ]
node001 [ UP ]
node002 [ UP ]
```

If some device is added to the cluster that blocks such pings, then the built-in ping can be replaced by the custom ping of the example, which relies on standard ICMP ping.

However, the replacement custom ping script need not actually use a variety of ping at all. It could be a script running web commands to query a chassis controller, asking if all its devices are up. The script simply has to provide an exit status compatible with expected ping behavior. Thus an exit status of 0 means all the devices are indeed up.

3.14.3 customremoteconsolescript

A custom remote console script can be used to run in the built-in remote console utility. This might be used, for example, to allow the administrator remote console access through a proprietary KVM switch client.

For example, a user may want to run a KVM console access script that is on the head node and with an absolute path on the head node of `/root/kvmaccesshack`. The script is to run on the console, and intended to be used for node `node001`, and takes the argument 1. This can then be set in cmsh as follows:

Example
3.15 Cluster Configuration Without Execution By CMDaemon

3.15.1 Cluster Configuration: The Bigger Picture

The configurations carried out in this chapter so far are based almost entirely on configuring nodes, via a CMDaemon front end (cmsh or Bright View), using CMDaemon to execute the change. Indeed, much of this manual is about this too because it is the preferred technique. It is preferred:

- because it is intended by design to be the easiest way to do common cluster tasks,
- and also generally keeps administration overhead minimal in the long run since it is CMDaemon rather than the system administrator that then takes care of tracking the cluster state.

There are however other cluster configuration techniques besides execution by CMDaemon. To get some perspective on these, it should be noted that cluster configuration techniques are always fundamentally about modifying a cluster so that it functions in a different way. The techniques can then for convenience be separated out into modification techniques that rely on CMDaemon execution and techniques that do not, as follows:

1. **Configuring nodes with execution by CMDaemon:** As explained, this is the preferred technique. The remaining techniques listed here should therefore usually only be considered if the task cannot be done with Bright View or cmsh.

2. **Replacing the node image:** The image on a node can be replaced by an entirely different one, so that the node can function in another way. This is covered in section 3.15.2. It can be claimed that since it is CMDaemon that selects the image, this technique should perhaps be classed as under item 1. However, since the execution of the change is really carried out by the changed image without CMDaemon running on the image, and because changing the entire image to implement a change of functionality is rather extreme, this technique can be given a special mention outside of CMDaemon execution.

3. **Using a FrozenFile directive:** Applied to a configuration file, this directive prevents CMDaemon from executing changes on that file for nodes. During updates, the frozen configuration may therefore need to be changed manually. The prevention of CMDaemon acting on that file prevents the standard cluster functionality that would run based on a fully CMDaemon-controlled cluster configuration. The FrozenFile directive is introduced in section 2.6.5, and covered in the configuration context in section 3.15.3.
4. **Using an initialize or finalize script:** This type of script is run during the initrd stage, much before CMDaemon on the regular node starts up. It is run if the functionality provided by the script is needed before CMDaemon starts up, or if the functionality that is needed cannot be made available later on when CMDaemon is started on the regular nodes. CMDaemon does not execute the functionality of the script itself, but the script is accessed and set on the initrd via a CMDaemon front end (Appendix E.2), and executed during the initrd stage. It is often convenient to carry out minor changes to configuration files inside a specific image in this way, as shown by the example in Appendix E.5. The initialize and finalize scripts are introduced in section 3.15.4.

5. **A shared directory:** Nodes can be configured to access and execute a particular software stored on a shared directory of the cluster. CMDaemon does not execute the functionality of the software itself, but is able to mount and share directories, as covered in section 3.10.

Finally, outside the stricter scope of cluster configuration adjustment, but nonetheless a broader way to modify how a cluster functions, and therefore mentioned here for more completeness, is:

6. **Software management:** the installation, maintenance, and removal of software packages. Standard post-installation software management based on repositories is covered in sections 11.2–11.6. Third-party software management from outside the repositories, for software that is part of Bright Cluster Manager is covered in Chapter 7 of the *Installation Manual*.

Third-party software that is not part of Bright Cluster Manager can be managed on the head node usually like to any other Linux system, under /opt or other recommended locations. If required by the other nodes, then the software should typically be set up by the administrator so that it can be accessed via a shared filesystem.

### 3.15.2 Making Nodes Function Differently By Image

#### Making All Nodes Function Differently By Image

To change the name of the image used for an entire cluster, for example after cloning the image and modifying it (section 3.15.2), the following methods can be used:

- in Bright View, via Cluster→Settings→Cluster name
- or in cmsh from within the base object of partition mode

A system administrator more commonly sets the software image on a per-category or per-node basis (section 3.15.2).

#### Making Some Nodes Function Differently By Image

For minor changes, adjustments can often be made to node settings via initialize and finalize scripts so that nodes or node categories function differently (section 3.15.4).

For major changes on a category of nodes, it is usually more appropriate to have nodes function differently from each other by simply carrying out image changes per node category with CMDaemon. Carrying out image changes per node is also possible. As usual, node settings override category settings.

#### Modifying images via cloning primitives for a node or category:

Setting a changed image for a category can be done as follows with cmsh:

1. The image on which the new one will be based is cloned. The cloning operation not only copies all the settings of the original (apart from the name), but also the data of the image:

   **Example**
3.15 Cluster Configuration Without Execution By CMDaemon

```
[root@bright82 ~]# cmsh
[bright82]# softwareimage
[bright82->softwareimage]# clone default-image imagetwo
[bright82->softwareimage+image[imagetwo]]# commit
images/default-image -> /cm/images/imagetwo
[bright82->softwareimage+image[imagetwo]]% commit
fault-image -> /cm/images/imagetwo
[bright82->softwareimage+image[imagetwo]]%
```

2. After cloning, the settings can be modified in the new object. For example, if the kernel needs to be
changed to suit nodes with different hardware, kernel modules settings are changed (section 5.3.2)
and committed. This creates a new image with a new ramdisk.

Other ways of modifying and committing the image for the nodes are also possible, as discussed
in sections 11.2–11.6 of this chapter.

3. The modified image that is to be used by the differently functioning nodes is placed in a new
category in order to have the nodes be able to choose the image. To create a new category easily, it
can simply be cloned. The image that the category uses is then set:

```
[bright82->softwareimage+image[imagetwo]]# category
[bright82->category]# clone default categorytwo
[bright82->category+category[categorytwo]]# set softwareimage imagetwo
[bright82->category+category[categorytwo]]# commit
[bright82->category+category[categorytwo]]%
```

4. • For just one node, or a few nodes, the node can be set from device mode to the new category
(which has the new image):

```
[bright82->category+category[categorytwo]]# device
[bright82->device]# use node099
[bright82->device+device[node099]]# set category categorytwo
[bright82->device+device[node099]]# commit; exit
```

• If there are many nodes, for example node100 sequentially up to node200, they can be set to
that category using a foreach loop like this:

**Example**

```
[bright82->device]# foreach -n node100..node200 (set category categorytwo)
[bright82->device+device]# commit
```

5. Rebooting restarts the nodes that are assigned to the new category with the new image.

**Modifying images via the cm/conf directory, for a category, node, or MAC address:** The preceding 5-
step method is understandable, but not very elegant. For Bright Cluster Manager 8.2 onwards, there is a
more structured and efficient way to make some nodes function differently by image. It relies on adding
the files in the image, via a target path that is specified in special configuration locations in the image. It
can be configured per node, but also per category and MAC address.

• For a category, the specification takes the form:

```
/cm/images/<image>/cm/conf/category/<category>/<target>
```

Thus, if some file on the node is to be placed so that on a running node it is at /path/to/some.file,
and this needs to be configured for an image default-image, and a category default, then it
would be placed at this location on the head node:

© Bright Computing, Inc.
Example

/cm/images/default-image/cm/conf/category/default/path/to/some.file

The file on the target node would be placed in the absolute directory /path/to/some.file

Multiple categories can be configured per image. Thus, for example, beside the default category, an additional gpu category can exist:

Example

/cm/images/default-image/cm/conf/category/gpu/path/to/some.file

Also, multiple files can be specified per category per image. Thus, beside the file some.file, an additional file some.other.file could be placed:

Example

/cm/images/default-image/cm/conf/category/gpu/path/to/some.file
/cm/images/default-image/cm/conf/category/gpu/path/to/some.other.file

• For a node, the configuration form is:
/cm/images/<image>/cm/conf/node/<node name>/<target>

An example for a node called node001 could then be:

Example

/cm/images/default-image/cm/conf/node/node001/path/to/some.file

• For a MAC address, the configuration form is:
/cm/images/<image>/cm/conf/node/<MAC address>/<target>

An example for a node with MAC address 00:aa:bb:cc:dd:ee could then be:

Example

/cm/images/default-image/cm/conf/node/00-aa-bb-cc-dd-ee/path/to/some.file

The copying of the specified files to the image is done just before the finalize stage of the node-installer (section 5.4.11) during node provisioning.

A common theme in Bright Cluster Manager is that node-level configuration overrides category-level configuration. In keeping with this behavior, a file configuration at category level could be applied to the many nodes in a category. And, a file configuration copy at node level (for a node that is in the category) overrides the category level value for just that particular node.

3.15.3 Making All Nodes Function Differently From Normal Cluster Behavior With FrozenFile

Configuration changes carried out by Bright View or cmsh often generate, restore, or modify configuration files (Appendix A).

However, sometimes an administrator may need to make a direct change (without using Bright View or cmsh) to a configuration file to set up a special configuration that cannot otherwise be done.

The FrozenFile directive to CMDaemon (Appendix C) applied to such a configuration file stops CMDaemon from altering the file. The frozen configuration file is generally applicable to all nodes and is therefore a possible way of making all nodes function differently from their standard behavior.

Freezing files is however best avoided, if possible, in favor of a CMDaemon-based method of configuring nodes, for the sake of administrative maintainability.
3.15.4 Adding Functionality To Nodes Via An *initialize* Or *finalize* Script

CMDaemon can normally be used to allocate different images per node or node category as explained in section 3.15.2. However, some configuration files do not survive a reboot (Appendix A), sometimes hardware issues can prevent a consistent end configuration, and sometimes drivers need to be initialized before provisioning of an image can happen. In such cases, an *initialize* or *finalize* script (sections 5.4.5, 5.4.11, and Appendix E.5) can be used to initialize or configure nodes or node categories.

These scripts are also useful because they can be used to implement minor changes across nodes:

Example

Supposing that some nodes with a particular network interface have a problem auto-negotiating their network speed, and default to 100Mbps instead of the maximum speed of 1000Mbps. Such nodes can be set to ignore auto-negotiation and be forced to use the 1000Mbps speed by using the \texttt{ETHTOOL_OPTS} configuration parameter in their network interface configuration file: \texttt{/etc/sysconfig/network-scripts/ifcfg-eth0} (or \texttt{/etc/sysconfig/network/ifcfg-eth0} in SUSE).

The \texttt{ETHTOOL_OPTS} parameter takes the options to the \texttt{"ethtool -s \textless device\textgreater"} command as options. The value of \texttt{\textless device\textgreater} (for example \texttt{eth0}) is specified by the filename that is used by the configuration file itself (for example \texttt{/etc/sysconfig/network-scripts/ifcfg-eth0}). The \texttt{ethtool} package is installed by default with Bright Cluster Manager. Running the command:

\begin{verbatim}
ethtool -s autoneg off speed 1000 duplex full
\end{verbatim}

turns out after some testing to be enough to reliably get the network card up and running at 1000Mbps on the problem hardware.

However, since the network configuration file is overwritten by node-installer settings during reboot, a way to bring persistence to the file setting is needed. One way to ensure persistence is to append the configuration setting to the file with a *finalize* script, so that it gets tagged onto the end of the configuration setting that the node-installer places for the file, just before the network interfaces are taken down again in preparation for *init*.

The script may thus look something like this for a Red Hat system:

\begin{verbatim}
#!/bin/bash

## node010..node014 get forced to 1000 duplex speed
if [[ $CMD_HOSTNAME = node01[0-4] ]]; then
  echo 'ETHTOOL_OPTS="speed 1000 duplex full"' >> /localdisk/etc/sysconfig/network-scripts/ifcfg-eth0
fi
\end{verbatim}

The method of enforcing an interface space just outlined is actually just for educational illustration, and is not a recommended method.

In practice, the recommended way to enforce an interface speed is to simply set it in the CMDaemon database. For example, for the boot interface of node001 it could be via the Bright View clickpath:

\begin{verbatim}
Devices\rightarrow Nodes[node001]\rightarrow Edit\rightarrow Settings\rightarrow Interfaces[BOOTIF]\rightarrow Edit\rightarrow Speed
\end{verbatim}

3.15.5 Examples Of Configuring Nodes With Or Without CMDaemon

A node or node category can often have its software configured in CMDaemon via Bright View or \texttt{cmsh}:

Example

Configuring a software for nodes using Bright View or \texttt{cmsh}: If the software under consideration is CUPS, then a node or node category can manage it from Bright View or \texttt{cmsh} as outlined in section 3.11.2.

A counterexample to this is:
Example

Configuring a software for nodes without using Bright View or cmsh\(^3\), using an image: Software images can be created with and without CUPS configured. Setting up nodes to load one of these two images via a node category is an alternative way of letting nodes run CUPS.

Whether node configuration for a particular functionality is done with CMDaemon, or directly with the software, depends on what an administrator prefers. In the preceding two examples, the first example, that is the one with Bright View or cmsh setting the CUPS service, is likely to be preferred over the second example, where an entire separate image must be maintained. A new category must also be created in the second case.

Generally, sometimes configuring the node via Bright Cluster Manager, and not having to manage images is better, sometimes configuring the software and making various images to be managed out of it is better, and sometimes only one of these techniques is possible anyway.

Configuring Nodes Using Bright View Or cmsh: Category Settings

When configuring nodes using Bright View or cmsh, configuring particular nodes from a node category to overrule the state of the rest of its category (as explained in section 2.1.3) is sensible for a small number of nodes. For larger numbers it may not be organizationally practical to do this, and another category can instead be created to handle nodes with the changes conveniently.

The CUPS service in the next two examples is carried out by implementing the changes via Bright View or cmsh acting on CMDaemon.

Example

Setting a few nodes in a category: If only a few nodes in a category are to run CUPS, then it can be done by enabling CUPs just for those few nodes, thereby overriding (section 2.1.3) the category settings.

Example

Setting many nodes to a category: If there are many nodes that are to be set to run CUPS, then a separate, new category can be created (cloning it from the existing one is easiest) and those many nodes are moved into that category, while the image is kept unchanged. The CUPS service setting is then set at category level to the appropriate value for the new category.

In contrast to these two examples, the software image method used in section 3.15.2 to implement a functionality such as CUPS would load up CUPS as configured in an image, and would not handle it via CMDaemon\(^3\). So, in section 3.15.2, software images prepared by the administrator are set for a node category. Since, by design, images are only selected for a category, a node cannot override the image used by the category other than by creating a new category, and using it with the new image. The administrative overhead of this can be inconvenient.

Administrators would therefore normally prefer letting CMDaemon track software functionality across nodes as in the last two examples, rather than having to deal with tracking software images manually. Indeed, the roles assignment option (section 2.1.5) is just a special pre-configured functionality toggle that allows CMDaemon to set categories or regular nodes to provide certain functions, typically by enabling services.

\(^3\)except to link nodes to their appropriate image via the associated category

© Bright Computing, Inc.
Aspects of power management in Bright Cluster Manager include:

- managing the main power supply to nodes through the use of power distribution units, baseboard management controllers, or CMDaemon
- monitoring power consumption over time
- setting CPU scaling governors for power-saving
- setting power-saving options in workload managers
- ensuring the passive head node can safely take over from the active head during failover (Chapter 14)
- allowing cluster burn tests to be carried out (Chapter 8 of the Installation Manual)

The ability to control power inside a cluster is therefore important for cluster administration, and also creates opportunities for power savings. This chapter describes the Bright Cluster Manager power management features.

In section 4.1 the configuration of the methods used for power operations is described. Section 4.2 then describes the way the power operations commands themselves are used to allow the administrator turn power on or off, reset the power, and retrieve the power status. It explains how these operations can be applied to devices in various ways.

Section 4.3 briefly covers monitoring power.

Section 4.4 describes how CMDaemon can set CPU defaults to control some of the CPU-dependent power consumption.

The integration of power saving with workload management systems is covered in the chapter on Workload Management (section 7.9).

### 4.1 Configuring Power Parameters

Several methods exist to control power to devices:

- Power Distribution Unit (PDU) based power control
- IPMI-based power control (for node devices only)
- Custom power control
- HP iLO-based power control (for node devices only)
- Dell DRAC-based power control (for node devices only)
- Cisco UCS CIMC-based power control (for node devices only)
- Redfish-based power control (for node devices only)
4.1.1 PDU-Based Power Control

For PDU-based power control, the power supply of a device is plugged into a port on a PDU. The device can be a node, but also anything else with a power supply, such as a switch. The device can then be turned on or off by changing the state of the PDU port.

To use PDU-based power control, the PDU itself must be a device in the cluster and be reachable over the network.

The `PowerDistributionUnits` menu option, accessible via the clickpath `Devices[device] → Settings` pane for each device plugged into the PDU, can then be used to configure the PDU ports that control the device. Figure 4.1 shows the menu option in the Settings window for a head node device, `bright82`, as accessed via the clickpath `Devices → Head Nodes[bright82] → Settings`.

![Figure 4.1: Power Section In The Head Node Settings](image)

If the menu option is clicked, then a subpane opens that allows each device plugged into the PDU to have PDU ports added and removed. For the APC brand of PDUs, the `Power control` property in the window of figure 4.1 should be set to `apc`, or the list of PDU ports is ignored by default. Overriding the default is described in section 4.1.3.

Since nodes may have multiple power feeds, there may be multiple PDU ports defined for a single device. The cluster management infrastructure takes care of operating all ports of a device in the correct order when a power operation is done on the device.

It is also possible for multiple devices to share the same PDU port. This is the case for example when *twin nodes* are used (i.e. two nodes sharing a single power supply). In this case, all power operations on one device apply to all nodes sharing the same PDU port.

If the PDUs defined for a node are not manageable, then the node’s baseboard management controllers (that is, IPMI/iLO and similar) are assumed to be inoperative and are therefore assigned an unknown state. This means that dumb PDUs, which cannot be managed remotely, are best not assigned to nodes in Bright Cluster Manager. Administrators wishing to use Bright Cluster Manager to record that a dumb PDU is assigned to a node can deal with it as follows:

- in Bright View the Notes field or the Userdefined1/Userdefined2 fields can be used.
• in cmsh the equivalent is accessible when using the node from device mode, and running “set notes”, “set userdefined1”, or “set userdefined2”.

For PDUs that are manageable:

• In cmsh, power-related options can be accessed from device mode, after selecting a device:

  Example

  [bright82] % device use node001
  [bright82->device[node001]] % show | grep -i power
  Custom power script argument
  Ipmi/iLO power reset delay 0
  Power control apc
  PowerDistributionUnits apc01:6 apc01:7

  The power status of a node can be accessed with:

  Example

  [bright82->device[node001]] % power status

  If the node is up and has one or more PDUs assigned to it, then the power status is one of ON, OFF, RESET, FAILED, or UNKNOWN:

<table>
<thead>
<tr>
<th>Power Status</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ON</td>
<td>Power is on</td>
</tr>
<tr>
<td>OFF</td>
<td>Power is off</td>
</tr>
<tr>
<td>RESET</td>
<td>Shows during the short time the power is off during a power reset. The reset is a hard power off for PDUs, but can be a soft or hard reset for other power control devices.</td>
</tr>
<tr>
<td>FAILED</td>
<td>Power status script communication failure.</td>
</tr>
<tr>
<td>UNKNOWN</td>
<td>Power status script timeout</td>
</tr>
</tbody>
</table>

4.1.2 IPMI-Based Power Control

IPMI-based power control relies on the baseboard management controller (BMC) inside a node. It is therefore only available for node devices. Blades inside a blade chassis typically use IPMI for power management. Section 3.7 describes setting up networking and authentication for IPMI/iLO/DRAC/CIMC/Redfish interfaces.

To carry out IPMI-based power control operations, the Power control property in figure 4.1 must be set to the IPMI interface through which power operations should be relayed. Normally this IPMI interface is configured to be ipmi0. Any list of configured APC PDU ports displayed in the GUI is ignored by default when the Power control property is not apc.

Example

Configuring power parameters settings for all the nodes using cmsh, with IPMI interfaces that are called ipmi0:
Example

Configuring power parameters settings for a node using cmsh with APC:

```bash
[mycluster]% device use node001
[...device[node001]% set powerdistributionunits apc01:6 apc01:7 apc01:8
[...device[node001]% get powerdistributionunits
  apc01:6 apc01:7 apc01:8
[...device[node001]% removefrom powerdistributionunits apc01:7
[...device[node001]% get powerdistributionunits
  apc01:6 apc01:8
[...device[node001]% set powercontrol apc
[...device[node001]% get powercontrol
  apc
[...device[node001]% commit
```

4.1.3 Combining PDU- and IPMI-Based Power Control

By default when nodes are configured for IPMI Based Power Control, any configured PDU ports are ignored. However, it is sometimes useful to change this behavior.

For example, in the CMDaemon configuration file directives in `/cm/local/apps/cmd/etc/cmd.conf` (introduced in section 2.6.2 and listed in Appendix C), the default value of `PowerOffPDUOutlet` is `false`. It can be set to `true` on the head node, and CMDaemon restarted to activate it.

With `PowerOffPDUOutlet` set to `true` it means that CMDaemon, after receiving an IPMI-based power off instruction for a node, and after powering off that node, also subsequently powers off the PDU port. Powering off the PDU port shuts down the BMC, which saves some additional power—typically a few watts per node. When multiple nodes share the same PDU port, the PDU port only powers off when all nodes served by that particular PDU port are powered off.

When a node has to be started up again the power is restored to the node. It is important that the node BIOS is configured to automatically power on the node when power is restored.

4.1.4 Custom Power Control

For a device which cannot be controlled through any of the standard existing power control options, it is possible to set a custom power management script. This is then invoked by the cluster management daemon on the head node whenever a power operation for the device is done.

Power operations are described further in section 4.2.

Using `custompowerscript`

To set a custom power management script for a device, the `powercontrol` attribute is set by the administrator to `custom` using either Bright View or cmsh, and the value of `custompowerscript` is specified by the administrator. The value for `custompowerscript` is the full path to an executable custom power management script on the head node(s) of a cluster.

A custom power script is invoked with the following mandatory arguments:

```
myscript <operation> <device>
```

where `<device>` is the name of the device on which the power operation is done, and `<operation>` is one of the following:

- ON
- OFF
- RESET
- STATUS

© Bright Computing, Inc.
On success a custom power script exits with exit code 0. On failure, the script exits with a non-zero exit-code.

**Using custompowerscriptargument**

The mandatory argument values for `<operation>` and `<device>` are passed to a custom script for processing. For example, in bash the positional variables `$1` and `$2` are typically used for a custom power script. A custom power script can also be passed a further argument value by setting the value of `custompowerscriptargument` for the node via `cmsh` or Bright View. This further argument value would then be passed to the positional variable `$3` in bash.

An example custom power script is located at `/cm/local/examples/cmd/custompower`. In it, setting `$3` to a positive integer delays the script via a `sleep` command by `$3` seconds.

An example that is conceivably more useful than a "sleep $3" command is to have a "wakeonlan $3" command instead. If the `custompowerscriptargument` value is set to the MAC address of the node, that means the MAC value is passed on to `$3`. Using this technique, the power operation ON can then carry out a Wake On LAN operation on the node from the head node.

Setting the `custompowerscriptargument` can be done like this for all nodes:

```bash
#!/bin/bash
for nodename in $(cmsh -c "device; foreach * (get hostname)")
do
  macad=`cmsh -c "device use $nodename; get mac"`
  cmsh -c "device use $nodename; set customscriptargument $macad; commit"
done
```

The preceding material usefully illustrates how `custompowerscriptargument` can be used to pass on arbitrary parameters for execution to a custom script.

However, the goal of the task can be achieved in a simpler and quicker way using the environment variables available in the cluster management daemon environment (section 3.3.1 of the *Developer Manual*). This is explained next.

**Using Environment Variables With custompowerscript**

Simplification of the steps needed for custom scripts in CMDaemon is often possible because there are values in the CMDaemon environment already available to the script. A line such as:

```
env > /tmp/env
```

added to the start of a custom script dumps the names and values of the environment variables to `/tmp/env` for viewing.

One of the names is `$CMD_MAC`, and it holds the MAC address string of the node being considered.

So, it is not necessary to retrieve a MAC value for `custompowerscriptargument` with a bash script as shown in the previous section, and then pass the argument via `$3` such as done in the command "wakeonlan $3". Instead, `custompowerscript` can simply call "wakeonlan $CMD_MAC" directly in the script when run as a power operation command from within CMDaemon.

### 4.1.5 Hewlett Packard iLO-Based Power Control

**iLO Configuration During Installation**

If "Hewlett Packard" is chosen as the node manufacturer during installation (section 3.3.6 of the *Installation Manual*), and the nodes have an iLO management interface, then Hewlett-Packard’s iLO management package, `hponcfg`, is installed by default on the nodes and head nodes.

**iLO Configuration After Installation**

If "Hewlett Packard" has not been specified as the node manufacturer during installation then it can be configured after installation as follows:

The `hponcfg` rpm package is normally obtained and upgraded for specific HP hardware from the HP website. Using an example of `hponcfg-3.1.1-0.noarch.rpm` as the package downloaded from the

© Bright Computing, Inc.
HP website, and to be installed, the installation can then be done on the head node, the software image, and in the node-installer as follows:

```
rpm -iv hponcfg-3.1.1-0.noarch.rpm
rpm --root /cm/images/default-image -iv hponcfg-3.1.1-0.noarch.rpm
rpm --root /cm/node-installer -iv hponcfg-3.1.1-0.noarch.rpm
```

To use iLO over all nodes, the following steps are done:

1. The iLO interfaces of all nodes are set up like the IPMI interfaces as outlined in section 4.1.2, using “set powercontrol ilo0” instead of “set powercontrol ipmi0”. Bright Cluster Manager treats HP iLO interfaces just like regular IPMI interfaces, except that the interface names are ilo0, ilo1,... instead of ipmi0, ipmi1...

2. The ilo_power.pl custom power script must be configured on all nodes. This can be done with a cmsh script. For example, for all nodes in the default category:

```
Example

[mycluster]% device foreach -c default (set custompowerscript /cm/ilo\ cal/apps/cmd/scripts/powerscripts/ilo_power.pl)
[mycluster]% device foreach -c default (set powercontrol custom)
[mycluster]% device commit
```

### 4.1.6 Dell drac-based Power Control

Dell drac configuration is covered on page 89.

### 4.1.7 Redfish-Based and CIMC-Based Power Control

Section 3.7 describes setting up networking and authentication for Redfish/CIMC, as well as for IPMI/iLO/DRAC interfaces.

### 4.2 Power Operations

#### 4.2.1 Power Operations Overview

**Main Power Operations**

Power operations may be carried out on devices from either Bright View or cmsh. There are four main power operations:

- **Power On**: power on a device
- **Power Off**: power off a device
- **Power Reset**: power off a device and power it on again after a brief delay
- **Power Status**: check power status of a device

**Scheduling-related Power Operations**

There are also scheduling-related power operations, which are currently (December 2018) only accessible via cmsh. Scheduling-related power operations are power operations associated with managing and viewing explicitly-scheduled execution.

Scheduled execution of power operations can be carried out explicitly via the --at, --after, -d, and -parallel-delay options. The scheduling-related power operations to manage and view such scheduled power operations are:

- **power wait**: Identifies the devices that have power operations that are in the waiting state, i.e. waiting to be carried out, and also outputs the number of operations that are waiting to be carried out.
4.2 Power Operations

- **power cancel**: Cancels an operation in the waiting state. The devices on which they should be cancelled can be specified.

- **power list**: Lists the power operations on the device and the states of the operations. Possible states for operations are:
  - waiting: waiting to be executed
  - busy: are being executed
  - canceled: have been canceled
  - done: have been executed

It is possible that power operations without an explicitly-scheduled execution time setting show up very briefly in the output of `power list` and `power wait`. However, the output displayed is almost always about the explicitly-scheduled power operations.

### 4.2.2 Power Operations With Bright View

In Bright View, executing the main power operations can be carried out as follows:

- via the menu dropdown for a node. For example:
  - for the head node, via the clickpath Devices → Head Nodes → Power
  - for a regular node via the clickpath Devices → Nodes → Power

- via the menu dropdown for a category or group. For example, for the default category, via the clickpath Grouping → Categories → Power

- via the *Actions* button. The *Actions* button is available when specific device has been selected. For example, for a head node bright82 the button can be seen via the clickpath Devices → Head Nodes[bright82] → Settings, as illustrated in figure 4.1. Clicking on the button makes power operation buttons available figure 4.2.

![Figure 4.2: Actions Button, Accessing The Power Operations](image)

### 4.2.3 Power Operations Through *cmsh*

Power operations on nodes can be carried out from within the `device` mode of *cmsh*, via the *power* command options.

© Bright Computing, Inc.
Powering On

Powering on can be carried out on a list of nodes (page 39). Powering on node001, and nodes from node018 to node033 (output truncated):

Example

```
[mycluster] $ device power -n node001,node018..node033 on
apc01:1 ............. [ ON ] node001
apc02:8 ............. [ ON ] node018
apc02:9 ............. [ ON ] node019
...
```

When a power operation is carried out on multiple devices, CMDaemon ensures that a 1 second delay occurs by default between successive devices. This helps avoid power surges on the infrastructure.

Delay Period Between Nodes

The delay period can be modified from within the device mode of cmsh, by using the -d|--delay option of the power command. For example, the preceding power command can be run with a shorter, 10ms delay with:

```
[mycluster] $ device power -n node001,node018..node033 -d 0.01 on
```

A 0-second delay (-d 0) should not be set for larger number of nodes, unless the power surge that this would cause has been taken into consideration.

Powering Up In Batches

Groups of nodes can be powered up “in batches”, according to power surge considerations. For example, to power up 3 racks at a time (“in batches of 3”), the -p|--parallel option is used:

Example

```
[mycluster] $ device power on -p 3 rack[01-12]
```

By default, there is a delay of 20s between batch commands. So, in the preceding example, there is a 20s pause before the each batch of the next three racks is powered up. For batch operation a delay of -d 0 is assumed, i.e. the nodes within in the rack are powered up without a built-in delay between the nodes of the rack.

Thread Use During Powering Up

The default number of threads that are started up to handle powering up of all the nodes is 32. If the hardware can cope with it, then it is possible to decrease startup time by increasing the default number of threads used to handle powerup, by editing the PowerThreadPoolSize advanced configuration directive in CMDaemon (page 643).

Powering Off Nodes

An example of powering off nodes is the following, where all nodes in the default category are powered off, with a 100ms delay between nodes (some output elided):

Example

```
[mycluster] $ device power off -c default -d 0.1
apc01:1 ............. [ OFF ] node001
apc01:2 ............. [ OFF ] node002
...
apc23:8 ............. [ OFF ] node953
```
4.2 Power Operations

Getting The Power Status
The power status command lists the status for devices:

Example

[mycluster]# device power status -g mygroup
apc01:3 ............. [ ON ] node003
apc01:4 ............. [ OFF ] node004

The power Command Help Text
The help text for the power command is:

[bright82->device]# help power
Name:
   power - Manipulate or retrieve power state of devices

Usage:

   power [OPTIONS] status
   power [OPTIONS] on
   power [OPTIONS] off
   power [OPTIONS] reset
   power [OPTIONS] list
   power [OPTIONS] cancel
   power [OPTIONS] wait <index>

Options:
- n, --nodes <node>
  List of nodes, e.g. node001..node015,node020..node028,node030 or
  `/some/file/containing/hostnames`
- g, --group <group>
  Include all nodes that belong to the node group, e.g. testnodes or test01,test03
- c, --category <category>
  Include all nodes that belong to the category, e.g. default or default,gpu
- r, --rack <rack>
  Include all nodes that are located in the given rack, e.g rack01 or
  rack01..rack04
- h, --chassis <chassis>
  Include all nodes that are located in the given chassis, e.g chassis01 or
  chassis03..chassis05
- e, --overlay <overlay>
  Include all nodes that are part of the given overlay, e.g overlay1 or
  overlayA,overlayC
- m, --image <image>
  Include all nodes that have the given image, e.g default-image or
  default-image,gpu-image
- t, --type <type>
  Type of devices, e.g node or virtualnode,cloudnode

© Bright Computing, Inc.
-i, --intersection
   Calculate the intersection of the above selections

-u, --union
   Calculate the union of the above selections

-l, --role role
   Filter all nodes that have the given role

-s, --status <status>
   Only run command on nodes with specified status, e.g. UP, "CLOSED|DOWN", "INST.*"

-b, --background
   Run in background, output will come as events

-d, --delay <seconds>
   Wait <seconds> between executing two sequential power commands. This option is ignored for the status command

-f, --force
   Force power command on devices which have been closed

-w, --overview
   Group all power operation results into an overview

-p, --parallel <number>
   Number of parallel option-items to be used per batch, default 0 (disabled)

--at <time>
   Execute the operation at the provided time

--after <seconds>
   Wait <seconds> before executing the operation

--parallel-delay <seconds>
   Wait <seconds> between executing the next batch of parallel commands, default 20s

--parallel-dry-run
   Only display the times at which operations will be executed, do not perform any power operations

--retry-count <number>
   Number of times to retry operation if it failed the first time (default 0)

--retry-delay <seconds>
   Delay between consecutive tries of a failed power operation (default 3s)

Examples:
  power status  Display power status for all devices or current device
  power on node001  Power on node001
  power on -n node00[1-2]  Power on node001 and node002
  power list  List all pending power operations
  power wait  List all power operation that can be waited for
4.3 Monitoring Power

Monitoring power consumption is important since electrical power is an important component of the total cost of ownership for a cluster. The monitoring system of Bright Cluster Manager collects power-related data from PDUs in the following metrics:

- `PDUBankLoad`: Phase load (in amperes) for one (specified) bank in a PDU
- `PDULoad`: Total phase load (in amperes) for one PDU

Chapter 12 on cluster monitoring has more on metrics and how they can be visualized.

4.4 CPU Scaling Governors

A cluster with CPU cores that run at a higher frequency consumes more power. A cluster administrator may therefore wish to implement schemes to control the CPU core frequencies, so that cluster power consumption is controlled.

4.4.1 The Linux Kernel And CPU Scaling Governors

In technology, a governor is the term used for a speed regulator. In computing, `CPU Scaling Governors` are power schemes that regulate what is commonly known as the CPU speed, or more precisely known as the CPU core clock frequency. The Linux kernel uses the `CPUFreq Governors` interface to implement these power schemes. The following governor values are currently commonly available to the interface:

<table>
<thead>
<tr>
<th>Governor</th>
<th>Policy Description For CPU Frequency</th>
</tr>
</thead>
<tbody>
<tr>
<td>performance</td>
<td>set to the highest allowed</td>
</tr>
<tr>
<td>userspace</td>
<td>set to that determined by root</td>
</tr>
<tr>
<td>ondemand</td>
<td>set according to demand, aggressively</td>
</tr>
<tr>
<td>conservative</td>
<td>set according to demand, non-aggressively</td>
</tr>
<tr>
<td>powersave</td>
<td>set to the lowest allowed</td>
</tr>
</tbody>
</table>

4.4.2 The Governor List According To `sysinfo`

Not all governors may be available for a particular CPU. Also, new governors may appear as new hardware is released. However, the hardware-defined list of governors that the kernel does detect as being available to a node `<hostname>` can always be seen in Bright Cluster Manager:

- in Bright View from within the CPU information, within the System Information pane of the host. For example, the clickpath for the head node host `bright82` is Devices → Head Nodes[bright82] → System Information, and the information is displayed in the PROCESSORS subpane under the AVAILABLEGOVERNORS column.
- in `cmsh`, by using the `sysinfo <hostname>` command, in device mode

The list can be displayed under `cmsh` in a form such as:

© Bright Computing, Inc.
Example

[bright82->device[bright82]]% sysinfo
System Information

----------------------------------------
Number of Cores 12
  Core 0 Six-Core AMD ... performance (ondemand, userspace)
  Core 1 Six-Core AMD ... performance (ondemand, userspace)

The governor in active operation for the cores is displayed outside the parentheses—in the example it is performance. The other two available, but inactive ones, are enclosed by parentheses—in the example they are ondemand and userspace. This list is essentially a hardware-defined list because its members depend only on the chip used. Which one of the members is active is however decided by a software setting for the governor value.

4.4.3 Setting The Governor

CPU scaling—Cool ‘n’ Quiet (AMD) or SpeedStep (Intel)—must be enabled in the BIOS for the governor to operate. Values for the governor may then be set in the BIOS. If a governor value is set in the BIOS, then the operating system cannot override it. If no value is set in the BIOS, then the value is set by the operating system.

On a running system, the cluster administrator can set the CPU governor value by using the CM-Daemon front ends as follows:

- In Bright View:
  - if the Node Categories resource is chosen, and a particular category is selected, then a Settings windows option for that category allows setting the Scaling governor to:
    * a single value.
    * multiple values. These can be set as a comma-separated list. This should not be confused with the hardware-defined list (section 4.4.2).
  - if the Nodes or Head Nodes resources are chosen, and a node selected from there, then its Settings window option allows its Scaling governor value to be set only to a single value.

- In cmsh:
  - if category mode, is used, and the category is chosen, then the scalinggovernor can be set to:
    * a single value.
    * multiple values. These can be set as a comma-separated list
  - if device mode is chosen, and a head or regular node selected from there, then its scalinggovernor value can be set only to a single value

If a list is set for scaling governors in Bright Cluster Manager, at category level, then the first value in the list is applicable to all the nodes in that category. If that fails for any of the nodes, for example if a node does not support this value, then the next value is attempted, and so on, until the node is assigned a governor value. This allows configuration according to preferred order to take place at category level.

As usual, configuration of a value at node level overrides the category value. However, in contrast with the setting that is possible at the category level, only a single value is possible at node level. This is because, if a specific node comes to the attention of the administrator for a custom governor value, then that node can be assigned the value decided by the administrator.

© Bright Computing, Inc.
4.5 Switch Configuration To Survive Power Downs

Besides the nodes and the BMC interfaces being configured for power control, it may be necessary to check that switches can handle power on and off network operations properly. Interfaces typically negotiate the link speed down to reduce power while still supporting Wake On Lan and other features. During such renegotiations the switch may lose connectivity to the node or BMC interface. This can happen if dynamic speed negotiation is disabled on the switch. Dynamic speed negotiation should therefore be configured to be on on the switch in order to reduce the chance that a node does not provision from a powered down state.

Example

[bright82->]% device use node001
[bright82->device[node001]]% sysinfo
System Information
-----------------------------------------------
... Number of Cores 12
Core 0 Six-Core AMD ... performance (ondemand, userspace)
Core 1 Six-Core AMD ... performance (ondemand, userspace)
... Core 11 Six-Core AMD ... performance (ondemand, userspace)

[bright82->device[node001]]% category use default
[bright82->category[default]]% set scalinggovernor ondemand,userspace,performance
[bright82->device[node001]]% device use node001
[bright82->device[node001]]% sysinfo
System Information
-----------------------------------------------
... Number of Cores 12
Core 0 Six-Core AMD ... ondemand (userspace, performance)
Core 1 Six-Core AMD ... ondemand (userspace, performance)
... Core 11 Six-Core AMD ... ondemand (userspace, performance)
This chapter covers node provisioning. Node provisioning is the process of how nodes obtain an image. Typically, this happens during their stages of progress from power-up to becoming active in a cluster, but node provisioning can also take place when updating a running node.

Section 5.1 describes the stages leading up to the loading of the kernel onto the node.

Section 5.2 covers configuration and behavior of the provisioning nodes that supply the software images.

Section 5.3 describes the configuration and loading of the kernel, the ramdisk, and kernel modules.

Section 5.4 elaborates on how the node-installer identifies and places the software image on the node in a 13-step process.

Section 5.5 explains node states during normal boot, as well node states that indicate boot problems.

Section 5.6 describes how running nodes can be updated, and modifications that can be done to the update process.

Section 5.7 explains how to add new nodes to a cluster so that node provisioning will work for these new nodes too. The Bright View and cmsh front ends for creating new node objects and properties in CMDaemon are described.

Section 5.8 describes troubleshooting the node provisioning process.

5.1 Before The Kernel Loads

Immediately after powering up a node, and before it is able to load up the Linux kernel, a node starts its boot process in several possible ways:

5.1.1 PXE Booting

By default, nodes boot from the network when using Bright Cluster Manager. This is called a network boot. On the x86 architectures it is known as a PXE boot (often pronounced as “pixie boot”). It is recommended as a BIOS setting for nodes. The head node runs a tftpd server from within xinetd under RHEL6, which supplies the boot loader from within the default software image (section 2.1.2) offered to nodes. For RHEL7 the tftpd server is managed by systemd.

The boot loader runs on the node and displays a menu (figure 5.1) based on loading a menu module within a configuration file. The default configuration files offered to nodes are located under /tftpboot/pxelinux.cfg/ on the head node. To implement changes in the files, CMDaemon may need to be restarted.

The default configuration files give instructions to the menu module of PXElinux. The instruction set used is documented at http://www.syslinux.org/wiki/index.php/Comboot/menu.c32, and includes the TIMEOUT, LABEL, MENU LABEL, DEFAULT, and MENU DEFAULT instructions.

The PXE TIMEOUT Instruction

During the display of the PXE boot menu, a selection can be made within a timeout period to boot the node in a several ways. Among the options are some of the install mode options (section 5.4.4). If no
selection is made by the user within the timeout period, then the AUTO install mode option is chosen by default.

In the PXE menu configuration files under \texttt{pxelinux.cfg/}, the default timeout of 5 seconds can be adjusted by changing the value of the “\texttt{TIMEOUT 50}” line. This value is specified in deciseconds.

\textbf{Example}

\begin{verbatim}
TIMEOUT 300  # changed timeout from 50 (=5 seconds)
\end{verbatim}

\textbf{The PXE LABEL And MENU LABEL Instructions}

\textbf{LABEL:} The menu configuration files under \texttt{pxelinux.cfg/} contain several multiline \texttt{LABEL} statements.

Each \texttt{LABEL} statement is associated with a kernel image that can be loaded from the PXE boot menu along with appropriate kernel options.

Each \texttt{LABEL} statement also has a text immediately following the \texttt{LABEL} tag. Typically the text is a description, such as \texttt{linux}, \texttt{main}, \texttt{RESCUE}, and so on. If the PXE menu module is not used, then tab completion prompting displays the list of possible text values at the PXE boot prompt so that the associated kernel image and options can be chosen by user intervention.

\textbf{MENU LABEL:} By default, the PXE menu module is used, and by default, each \texttt{LABEL} statement also contains a \texttt{MENU LABEL} instruction. Each \texttt{MENU LABEL} instruction also has a text immediately following the \texttt{MENU LABEL} tag. Typically the text is a description, such as \texttt{AUTO}, \texttt{RESCUE} and so on (figure 5.1). Using the PXE menu module means that the list of the \texttt{MENU LABEL} text values is displayed when the PXE boot
5.1 Before The Kernel Loads

The menu is displayed, so that the associated kernel image and options can conveniently be selected by user intervention.

The PXE DEFAULT And MENU DEFAULT Instructions

DEFAULT: If the PXE menu module is not used and if no MENU instructions are used, and if there is no user intervention, then setting the same text that follows a LABEL tag immediately after the DEFAULT instruction, results in the associated kernel image and its options being run by default after the timeout.

By default, as already explained, the PXE menu module is used. In particular it uses the setting: DEFAULT menu.c32 to enable the menu.

MENU DEFAULT: If the PXE menu module is used and if MENU instructions are used, and if there is no user intervention, then setting a MENU DEFAULT tag as a line within the multiline LABEL statement results in the kernel image and options associated with that LABEL statement being loaded by default after the timeout.

The CMDaemon PXE LABEL Setting For Specific Nodes

The MENU DEFAULT value by default applies to every node using the software image that the PXE menu configuration file under pxelinux.cfg/ is loaded from. To override its application on a per-node basis, the value of PXE LABEL can be set for each node.

• Some simple examples of overriding the default MENU DEFAULT value are as follows:

   For example, using cmsh:

   Example

   [root@bright82 ~]# cmsh
   [bright82]$ device use node001
   [bright82->device[node001]]% set pxelabel MEMTEST ; commit

   Carrying it out for all nodes in the default category can be done, for example, with:

   Example

   [root@bright82 ~]# cmsh
   [bright82]$ device
   [bright82->device]% foreach -c default (set pxelabel MEMTEST)
   [bright82->device*]% commit

   The value of pxelabel can be cleared with:

   Example

   [root@bright82 ~]# cmsh -c "device; foreach -c default (clear pxelabel); commit"

   In Bright View, the PXE label can be set from the Settings window for a node (figure 5.2).

• A more complicated example of overriding the default MENU DEFAULT value now follows. Although it helps in understanding how PXE labels can be used, it can normally be skipped because the use case for it is unlikely, and the details are involved.

   In this example, pxelabel is set by the administrator via Bright View or cmsh to localdrive. This will then set the node to boot from the first local drive and not the node-installer. This is a setting that is discouraged since it usually makes node management harder, but it can be used by

© Bright Computing, Inc.
administrators who do not wish to answer any prompt during node boot, and also want the node
drives to have no risk of being overwritten by the actions of the node-installer, and also want the
system to be up and running quite fully, even if not necessarily provisioned with the latest image
from the head node.

Here, the overwriting-avoidance method relies on the nodes being associated with a configuration
file under pxelinux.cfg at the time that the localdrive setting is done. However, nodes that are
unidentified, or are identified later on, will have their MENU DEFAULT value still set to a default
pxelabel value set in the files under /tftpboot/pxelinux.cfg/, which is the value linux by
default, and which is associated with a code block in that file with the label LABEL linux. To make
such (as yet) unidentified nodes boot to a localdrive setting instead, requires modifying the files
under /tftpboot/pxelinux.cfg/, so that the MENU DEFAULT line is associated with the code block
of LABEL localdrive rather than the code block of LABEL linux.

There are two methods other than using the preceding pxelabel method to deal with the risk of
overwriting. Unlike the pxelabel method however, these methods can interrupt node booting, so
that the node does not progress to being fully up until the administrator takes further action:

1. If it is acceptable that the administrator manually enters a confirmation as part of the boot
process when a possible overwrite risk is found, then the datanode method (section 5.4.4)
can be used.
2. If it is acceptable that the boot process halts on detecting a possible overwrite risk, then the
XML assertions method (Appendix D.11) is recommended.

**Changing The Install Mode Or Default Image Offered To Nodes**
The selections offered by the PXE menu are pre-configured by default so that the AUTO menu option by
default loads a kernel, runs the AUTO install mode, and eventually the default-image software image is
provisioned.

Normally administrators should not be changing the install mode, kernel, or kernel options in the
PXE menu configuration files under pxelinux.cfg/.
More on changing the install mode is given in section 5.4.4. More on changing software images, image package management, kernels, and kernel options, is to be found in Chapter 11.

5.1.2 IPXE Booting From A Disk Drive
Also by default, on disked nodes, iPXE software is placed on the drive during node installation. If the boot instructions from the BIOS for PXE booting fail, and if the BIOS instructions are that a boot attempt should then be made from the hard drive, it means that a PXE network boot attempt is done again, as instructed by the bootable hard drive. This can be a useful fallback option that works around certain BIOS features or problems.

5.1.3 IPXE Booting Using InfiniBand
On clusters that have InfiniBand hardware, it is normally used for data transfer as a service after the nodes have fully booted up (section 3.6). InfiniBand can also be used for PXE booting (described here) and used for node provisioning (section 5.3.3). However these uses are not necessary, even if InfiniBand is used for data transfer as a service later on, because booting and provisioning is available over Ethernet by default. This section (about boot over InfiniBand) may therefore safely be skipped when first configuring a cluster.

Booting over InfiniBand via PXE is enabled by carrying out these 3 steps:

1. Making the Bright Cluster Manager aware that nodes are to be booted over InfiniBand. This can be done during the initial installation on the head node by marking the option “Allow booting over InfiniBand” (figure 3.15 of the Installation Manual). Alternatively, if the cluster is already installed, then node booting (section 3.2.3, page 70) can be set from cmsh as follows:

(a) From cmsh’s network mode: If the InfiniBand network name is ibnet, then a cmsh command that will set it is:

```
cmsh -c "network; set ibnet nodebooting yes; commit"
```

(b) From Bright View: The Settings window for the InfiniBand network, for example ibnet, can be accessed from the Networking resource via the clickpath Networking→Networks[ibnet]→Edit→Settings (this is similar to figure 3.5, but for ibnet). The Node booting option for ibnet is then enabled and saved.

If the InfiniBand network does not yet exist, then it must be created (section 3.2.2). The recommended default values used are described in section 3.6.3. The MAC address of the interface in CMDaemon defaults to using the GUID of the interface.

The administrator should also be aware that the interface from which a node boots, (conveniently labeled BOOTIF), must not be an interface that is already configured for that node in CMDaemon. For example, if BOOTIF is the device ib0, then ib0 must not already be configured in CMDaemon. Either BOOTIF or the ib0 configuration should be changed so that node installation can succeed. It is recommended to set BOOTIF to eth0 if the ib0 device should exist.

2. Flashing iPXE onto the InfiniBand HCAs. (The ROM image is obtained from the HCA vendor).

3. Configuring the BIOS of the nodes to boot from the InfiniBand HCA. For PXE booting over Omni-Path, older Omni-Path cards may need to have UEFI firmware installed on them, and the only supported boot mode is then UEFI.

All MAC addresses become invalid for identification purposes when changing from booting over Ethernet to booting over InfiniBand.

Administrators who enable iPXE booting almost always wish to provision over InfiniBand too. Configuring provisioning over InfiniBand is described in section 5.3.3.
5.1.4 Using PXE To Boot From The Drive
Besides PXE booting from only the network, a node can also be configured via PXE to step over to using its own drive to start booting and get to the stage of loading up its kernel entirely from its drive, just like a normal standalone machine. This can be done by setting PXE LABEL to `localdrive` (page 147).

5.1.5 Network Booting Without PXE On The ARMv8 Architecture
ARMv8 nodes use a network boot implementation that differs slightly from the x86 PXE boot implementation. The actual firmware that starts up on ARMv8 nodes depends on the environment the hardware is in. Networking is then started by the firmware and the network requests what to boot. The head node however then sends out a GRUB binary instead of an iPXE binary. The GRUB binary then runs on the regular node, and fetches the kernel and initrd via TFTP. The node installer then runs on the nodes and follows the the same steps as in the x86 process.

5.1.6 The Boot Role
The action of providing a boot image to a node via DHCP and TFTP is known as providing node booting. Node provisioning (section 5.2), on the other hand, is about provisioning the node with the rest of the node image.

Roles in general are introduced in section 2.1.5. The boot role is one such role that can be assigned to a regular node. The boot role configures a regular node so that it can then provide node booting. The role cannot be assigned or removed from the head node—the head node always has a boot role.

The boot role is assigned by administrators to regular nodes if there is a need to cope with the scaling limitations of TFTP and DHCP. TFTP and DHCP services can be overwhelmed when there are large numbers of nodes making use of them during boot. An example of the scaling limitations may be observed, for example, when, during the powering up and network booting attempts of a large number of regular nodes from the head node, it turns out that random different regular nodes are unable to boot, typically due to network effects.

One implementation of boot role assignment might therefore be, for example, to have a several groups of racks, with each rack in a subnet, and with one regular node in each subnet that is assigned the boot role. The boot role regular nodes would thus take the DHCP and TFTP load off the head node and onto themselves for all the nodes in their associated subnet, so that all nodes of the cluster are then able to boot without networking issues.

5.2 Provisioning Nodes
The action of transferring the software image to the nodes is called node provisioning, and is done by special nodes called the provisioning nodes. More complex clusters can have several provisioning nodes configured by the administrator, thereby distributing network traffic loads when many nodes are booting.

Creating provisioning nodes is done by assigning a provisioning role to a node or category of nodes. Similar to how the head node always has a boot role (section 5.1.6), the head node also always has a provisioning role.

5.2.1 Provisioning Nodes: Configuration Settings
The provisioning role has several parameters that can be set:
### 5.2 Provisioning Nodes

<table>
<thead>
<tr>
<th>Property</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>allImages</strong></td>
<td>The following values decide what images the provisioning node provides:</td>
</tr>
<tr>
<td></td>
<td>• onlocaldisk: all images on the local disk, regardless of any other parameters set</td>
</tr>
<tr>
<td></td>
<td>• onsharedstorage: all images on the shared storage, regardless of any other parameters set</td>
</tr>
<tr>
<td></td>
<td>• no (the default): only images listed in the localimages or sharedimages parameters, described next</td>
</tr>
<tr>
<td><strong>localimages</strong></td>
<td>A list of software images on the local disk that the provisioning node accesses and provides. The list is used only if allImages is “no”.</td>
</tr>
<tr>
<td><strong>sharedimages</strong></td>
<td>A list of software images on the shared storage that the provisioning node accesses and provides. The list is used only if allImages is “no”</td>
</tr>
<tr>
<td><strong>Provisioning slots</strong></td>
<td>The maximum number of nodes that can be provisioned in parallel by the provisioning node. The optimum number depends on the infrastructure. The default value is 10, which is safe for typical cluster setups. Setting it lower may sometimes be needed to prevent network and disk overload.</td>
</tr>
<tr>
<td><strong>nodegroups</strong></td>
<td>A list of node groups (section 2.1.4). If set, the provisioning node only provisions nodes in the listed groups. Conversely, nodes in one of these groups can only be provisioned by provisioning nodes that have that group set. Nodes without a group, or nodes in a group not listed in nodegroups, can only be provisioned by provisioning nodes that have no nodegroups values set. By default, the nodegroups list is unset in the provisioning nodes. The nodegroups setting is typically used to set up a convenient hierarchy of provisioning, for example based on grouping by rack and by groups of racks.</td>
</tr>
</tbody>
</table>

A provisioning node keeps a copy of all the images it provisions on its local drive, in the same directory as where the head node keeps such images. The local drive of a provisioning node must therefore have enough space available for these images, which may require changes in its disk layout.

#### 5.2.2 Provisioning Nodes: Role Setup With cmsh

In the following cmsh example the administrator creates a new category called misc. The default category default already exists in a newly installed cluster.

The administrator then assigns the role called provisioning, from the list of available assignable roles, to nodes in the misc category. After the assign command has been typed in, but before entering the command, tab-completion prompting can be used to list all the possible roles. Assignment creates an association between the role and the category. When the assign command runs, the shell drops into the level representing the provisioning role.

If the role called provisioning were already assigned, then the use provisioning command would drop the shell into the provisioning role, without creating the association between the role and the category.

As an aside from the topic of provisioning, from an organizational perspective, other assignable roles include monitoring, storage, and failover.

Once the shell is within the role level, the role properties can be edited conveniently.

© Bright Computing, Inc.
For example, the nodes in the misc category assigned the provisioning role can have default-image set as the image that they provision to other nodes, and have 20 set as the maximum number of other nodes to be provisioned simultaneously (some text is elided in the following example):

**Example**

```
[bright82] % category add misc
[bright82->category[misc*]] % roles
[bright82->category[misc*]->roles] % assign provisioning
[bright82...->roles*{provisioning*}] % set allimages no
[bright82...->roles*{provisioning*}] % set localimages default-image
[bright82...->roles*{provisioning*}] % set provisioningslots 20
[bright82...->roles*{provisioning*}] % show
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>All Images</td>
<td>no</td>
</tr>
<tr>
<td>Include revisions of local images</td>
<td>yes</td>
</tr>
<tr>
<td>Local images</td>
<td>default-image</td>
</tr>
<tr>
<td>Name</td>
<td>provisioning</td>
</tr>
<tr>
<td>Nodegroups</td>
<td></td>
</tr>
<tr>
<td>Provisioning associations</td>
<td>&lt;0 internally used&gt;</td>
</tr>
<tr>
<td>Revision</td>
<td></td>
</tr>
<tr>
<td>Type</td>
<td>ProvisioningRole</td>
</tr>
<tr>
<td>Provisioning slots</td>
<td>20</td>
</tr>
</tbody>
</table>

```
[bright82->category[misc*]->roles*{provisioning*}] % commit
[bright82->category[misc*]->roles*{provisioning*}] %
```

Assigning a provisioning role can also be done for an individual node instead, if using a category is deemed overkill:

**Example**

```
[bright82] % device use node001
[bright82->device[node001]] % roles
[bright82->device[node001]->roles] % assign provisioning
[bright82->device[node001*]->roles*{provisioning*}] %
```

A role change configures a provisioning node, but does not directly update the provisioning node with images. After carrying out a role change, Bright Cluster Manager runs the updateprovisioners command described in section 5.2.4 automatically, so that regular images are propagated to the provisioners. The propagation can be done by provisioners themselves if they have up-to-date images. CMDaemon tracks the provisioning nodes role changes, as well as which provisioning nodes have up-to-date images available, so that provisioning node configurations and regular node images propagate efficiently. Thus, for example, image update requests by provisioning nodes take priority over provisioning update requests from regular nodes.

### 5.2.3 Provisioning Nodes: Role Setup With Bright View

The provisioning configuration outlined in cmsh mode in section 5.2.2 can be done via Bright View too, as follows:

A misc category can be added via the clickpath:

Grouping→Categories→Add→Settings→<name>

The node category should be given a name misc (figure 5.3), and saved:
Figure 5.3: Bright View: Adding A misc Category

The Roles window can then be opened from within the misc category. To add a role, the Add button in the Roles window is clicked. A scrollable list of available roles is then displayed, (figure 5.4):

Figure 5.4: Bright View: Setting A provisioning Role

After selecting a role, then navigating via the Back buttons to the Settings menu of figure 5.3, the role can be saved using the Save button there.

The role has properties which can be edited (figure 5.5):
The **Provisioning slots** setting decides, for example, how many images can be supplied simultaneously from the provisioning node, while the **Images** and **All images** settings decide what images the provisioning node supplies. As before, the setting can be saved using the **Save** button of figure 5.3.

The images offered by the provisioning role should not be confused with the software image setting of the **misc** category itself, which is the image the provisioning node requests for itself from the category.

### 5.2.4 Provisioning Nodes: Housekeeping

The head node does housekeeping tasks for the entire provisioning system. Provisioning is done on request for all non-head nodes on a first-come, first-serve basis. Since provisioning nodes themselves, too, need to be provisioned, it means that to cold boot an entire cluster up quickest, the head node should be booted and be up first, followed by provisioning nodes, and finally by all other non-head nodes. Following this start-up sequence ensures that all provisioning services are available when the other non-head nodes are started up.

Some aspects of provisioning housekeeping are discussed next:

**Provisioning Node Selection**

When a node requests provisioning, the head node allocates the task to a provisioning node. If there are several provisioning nodes that can provide the image required, then the task is allocated to the provisioning node with the lowest number of already-started provisioning tasks.

**Limiting Provisioning Tasks With MaxNumberOfProvisioningThreads**

Besides limiting how much simultaneous provisioning per provisioning node is allowed with **Provisioning Slots** (section 5.2.1), the head node also limits how many simultaneous provisioning tasks are allowed to run on the entire cluster. This is set using the `MaxNumberOfProvisioningThreads` directive in the head node’s CMDaemon configuration file, `/etc/cmd.conf`, as described in Appendix C.
Provisioning Tasks Deferral and Failure

A provisioning request is deferred if the head node is not able to immediately allocate a provisioning node for the task. Whenever an ongoing provisioning task has finished, the head node tries to re-allocate deferred requests.

A provisioning request fails if an image is not transferred. 5 retry attempts at provisioning the image are made in case a provisioning request fails.

A provisioning node that is carrying out requests, and which loses connectivity, has its provisioning requests remain allocated to it for 180 seconds from the time that connectivity was lost. After this time the provisioning requests fail.

Provisioning Role Change Notification With updateprovisioners

The updateprovisioners command can be accessed from the softwareimage mode in cmsh. It can also be accessed from Bright View, via the clickpath Provisioning → Provisioning requests → Update provisioning nodes.

In the examples in section 5.2.2, changes were made to provisioning role attributes for an individual node as well as for a category of nodes. This automatically ran the updateprovisioners command.

The updateprovisioners command runs automatically if CMDaemon is involved during software image changes or during a provisioning request. If on the other hand, the software image is changed outside of the CMDaemon front ends (Bright View and cmsh), for example by an administrator adding a file by copying it into place from the bash prompt, then updateprovisioners should be run manually to update the provisioners.

In any case, if it is not run manually, it is scheduled to run every midnight by default.

When the default updateprovisioners is invoked manually, the provisioning system waits for all running provisioning tasks to end, and then updates all images located on any provisioning nodes by using the images on the head node. It also re-initializes its internal state with the updated provisioning role properties, i.e. keeps track of what nodes are provisioning nodes.

The default updateprovisioners command, run with no options, updates all images. If run from cmsh with a specified image as an option, then the command only does the updates for that particular image. A provisioning node undergoing an image update does not provision other nodes until the update is completed.

Example

[bright82]# softwareimage updateprovisioners
Provisioning nodes will be updated in the background.

Sun Dec 12 13:45:09 2010 bright82: Starting update of software image(s)\n  provisioning node(s). (user initiated).
[bright82]# softwareimage updateprovisioners [bright82]%
Sun Dec 12 13:45:41 2010 bright82: Updating image default-image on prov\n isioning node node001.
[bright82]%
Sun Dec 12 13:46:00 2010 bright82: Updating image default-image on prov\n isioning node node001 completed.
Sun Dec 12 13:46:00 2010 bright82: Provisioning node node001 was updated
Sun Dec 12 13:46:00 2010 bright82: Finished updating software image(s) \n  on provisioning node(s).

Provisioning Role Draining And Undraining Nodes With drain, undrain

The drain and undrain commands to control provisioning nodes are accessible from within the softwareimage mode of cmsh.

If a node is put into a drain state, then all currently active provisioning requests continue until they are completed. However, the node is not assigned any further pending requests, until the node is put back into an undrain state.
Example

[bright82->softwareimage] % drain -n master
Nodes drained
[bright82->softwareimage] % provisioningstatus
Provisioning subsystem status
Pending request: node001, node002
Provisioning node status:
+ bright82
  Slots: 1 / 10
  State: draining
  Active nodes: node003
  Up to date images: default-image
[bright82->softwareimage] % provisioningstatus
Provisioning subsystem status
Pending request: node001, node002
Provisioning node status:
+ bright82
  Slots: 0 / 10
  State: drained
  Active nodes: none
  Up to date images: default-image

To drain all nodes at once, the --role option can be used, with provisioning role as its value. All pending requests then remain in the queue, until the nodes are undrained again.

Example

[bright82->softwareimage] % drain --role provisioning
...Time passes. Pending
requests stay in the queue. Then
admin undrains it...
[bright82->softwareimage] % undrain --role provisioning

Provisioning Node Update Safeguards And
provisioningnodeautoupdatetimelimit

The updateprovisioners command is subject to safeguards that prevent it running too frequently. The minimum period between provisioning updates can be adjusted with the parameter provisioning-nodeautoupdatetimelimit, which has a default value of 300s.

Exceeding the timeout does not by itself trigger an update to the provisioning node.

When the head node receives a provisioning request, it checks if the last update of the provisioning nodes is more than the timeout period. If true, then an update is triggered to the provisioning node. The update is disabled if the timeout is set to zero (false).

The parameter can be accessed and set within cmsh from partition mode:

Example

[root@bright82 ]# cmsh
[bright82] % partition use base
[bright82->partition[base]] % get provisioningnodeautoupdatetimelimit
[bright82->partition[base]] % 300
[bright82->partition[base]] % set provisioningnodeautoupdatetimelimit 0
[bright82->partition[base]] % commit

Within Bright View the parameter is accessible via the clickpath:

To prevent provisioning an image to the nodes, the locked state (section 5.4.7) can be used. A locked state defers the provisioning request, until the image is unlocked once more.
5.3 The Kernel Image, Ramdisk And Kernel Modules

A *software image* is a complete Linux filesystem that is to be installed on a non-head node. Chapter 11 describes images and their management in detail.

The head node holds the head copy of the software images. Whenever files in the head copy are changed using CMDaemon, the changes automatically propagate to all provisioning nodes via the `updateprovisioners` command (section 5.2.4).

5.3.1 Booting To A “Good State” Software Image

When nodes boot from the network in simple clusters, the head node supplies them with a *known good state* during node start up. The known good state is maintained by the administrator and is defined using a software image that is kept in a directory of the filesystem on the head node. Supplementary filesystems such as `/home` are served via NFS from the head node by default.

For a diskless node the known good state is copied over from the head node, after which the node becomes available to cluster users.

For a disked node, by default, the hard disk contents on specified local directories of the node are checked against the known good state on the head node. Content that differs on the node is changed to that of the known good state. After the changes are done, the node becomes available to cluster users.

Each software image contains a Linux kernel and a ramdisk. These are the first parts of the image that are loaded onto a node during early boot. The kernel is loaded first. The ramdisk is loaded next, and contains driver modules for the node’s network card and local storage. The rest of the image is loaded after that, during the node-installer stage (section 5.4).

5.3.2 Selecting Kernel Driver Modules To Load Onto Nodes

Kernel modules can be managed in `softwareimage` mode (using an image), in `category` mode (using a category), or in `device` mode (using a node), as indicated by the following `cmsh` tree view of a newly-installed cluster with default values:

```
cmsh
|-- category[default]
  |-- kernelmodules
...  
|-- device[node001]
  |-- kernelmodules
...  
|-- softwareimage[default-image]
  |-- kernelmodules
...  
```

As is usual in Bright Cluster Manager, if there are values specified at the lower levels in the hierarchy, then their values override the values set higher up in the hierarchy. For example, modules specified at node level override modules specified at category or software image level. Similarly modules specified at the category level override whatever is specified at the software image level. The cluster administrators should be aware that “override” for kernel modules appended to a kernel image means that any kernel modules defined at a higher level are totally ignored—the modules from a lower level exclude the modules at the higher level. A misconfiguration of kernel modules in the lower levels can thus prevent the node from starting up.

Modules are normally just set at `softwareimage` level, using `cmsh` or Bright View.

**Kernel Driver Modules With cmsh**

In `cmsh`, the modules that are to go on the ramdisk can be placed using the `kernelmodules` submode of the `softwareimage` mode. The order in which they are listed is the attempted load order.

Within the `kernelmodules` submode, the `import` command can be used to import the kernel modules list from a software image, from a node, or from a category, replacing the original kernel modules list.
Whenever a change is made via the kernelmodules submode to the kernel module selection of a software image, CMDaemon automatically runs the createramdisk command. The createramdisk command regenerates the ramdisk inside the initrd image and sends the updated image to all provisioning nodes, to the image directory, set by default to /cm/images/default-image/boot/. The original initrd image is saved as a file with suffix ".orig" in that directory. An attempt is made to generate the image for all software images that CMDaemon is aware of, regardless of category assignment, unless the image is protected from modification by CMDaemon with a FrozenFile directive (Appendix C).

The createramdisk command can also be run manually from within the softwareimage mode.

**Kernel Driver Modules With Bright View**

In Bright View the selection of kernel modules for a particular image, is carried out through the Software images resource, and then choosing the Kernel modules menu option of that image. For example, for the image default-image, clickpath Provisioning→Software images[default-image]→Edit→Settings→Kernel modules, leading to figure 5.6:

![Figure 5.6: Bright View: Selecting Kernel Modules For Software Images](image)

New kernel modules can be added using the Add button, existing kernel modules can be removed using the Delete button, and kernel module parameters can be edited using the Edit button.

**Manually Regenerating A Ramdisk**

Regenerating a ramdisk manually via cmsh or Bright View is useful if the kernel or modules have changed without using CMDaemon. For example, after running a YUM update which has modified the kernel or modules of the nodes (section 11.3). In such a case, the distribution would normally update the ramdisk on the machine, but this is not done for the extended ramdisk for nodes in Bright Cluster Manager. Not regenerating the Bright Cluster Manager ramdisk for nodes after such an update means the nodes may fail on rebooting during the loading of the ramdisk (section 5.8.4).

An example of regenerating the ramdisk is seen in section 5.8.5.

**Implementation Of Kernel Driver Via Ramdisk Or Kernel Parameter**

Sometimes, testing or setting a kernel driver as a kernel parameter may be more convenient. How to do that is covered in section 11.3.4.
5.3 The Kernel Image, Ramdisk And Kernel Modules

5.3.3 InfiniBand Provisioning

On clusters that have InfiniBand hardware, it is normally used for data transfer as a service after the nodes have fully booted up (section 3.6). It can also be used for PXE booting (section 5.1.3) and for node provisioning (described here), but these are not normally a requirement. This section (about InfiniBand node provisioning) may therefore safely be skipped in almost all cases when first configuring a cluster.

During node start-up on a setup for which InfiniBand networking has been enabled, the init process runs the rdma script. For SLES and distributions based on versions prior to Red Hat 6, the openib script is used instead of the rdma script. The script loads up InfiniBand modules into the kernel. When the cluster is finally fully up and running, the use of InfiniBand is thus available for all processes that request it. Enabling InfiniBand is normally set by configuring the InfiniBand network when installing the head node, during the Additional Network Configuration screen (figure 3.15 of the Installation Manual).

Provisioning nodes over InfiniBand is not implemented by default, because the init process, which handles initialization scripts and daemons, takes place only after the node-provisioning stage launches. InfiniBand modules are therefore not available for use during provisioning, which is why, for default kernels, provisioning in Bright Cluster Manager is done via Ethernet.

Provisioning at the faster InfiniBand speeds rather than Ethernet speeds is however a requirement for some clusters. To get the cluster to provision using InfiniBand requires both of the following two configuration changes to be carried out:

1. configuring InfiniBand drivers for the ramdisk image that the nodes first boot into, so that provisioning via InfiniBand is possible during this pre-init stage
2. defining the provisioning interface of nodes that are to be provisioned with InfiniBand. It is assumed that InfiniBand networking is already configured, as described in section 3.6.

   The administrator should be aware that the interface from which a node boots, (conveniently labeled BOOTIF), must not be an interface that is already configured for that node in CMDaemon. For example, ifBOOTIF is the device ib0, then ib0 must not already be configured in CMDaemon. Either BOOTIF or the ib0 configuration should be changed so that node installation can succeed.

   How these two changes are carried out is described next:

InfiniBand Provisioning: Ramdisk Image Configuration

An easy way to see what modules must be added to the ramdisk for a particular HCA can be found by running rdma (or openibd), and seeing what modules do load up on a fully booted regular node.

   One way to do this is to run the following lines as root:

   [root@bright82 ~] # { service rdma stop; lsmod | cut -f1 -d" " ; }>/tmp/a
   [root@bright82 ~] # { service rdma start; lsmod | cut -f1 -d" " ; }>/tmp/b

   The rdma service in the two lines should be replaced by openibd service instead when using SLES, or distributions based on versions of Red Hat prior to version 6.

   The first line stops the InfiniBand service, just in case it is running, in order to unload its modules, and then lists the modules on the node.

   The second line starts the service, so that the appropriate modules are loaded, and then lists the modules on the node again. The output of the first step is stored in a file a, and the output from the second step is stored in a file b.

   Running diff on the output of these two steps then reveals the modules that get loaded. For rdma, the output may display something like:

   Example

   [root@bright82 ~] # diff /tmp/a /tmp/b
      1,3c1
      < Unloading OpenIB kernel modules:

© Bright Computing, Inc.
As suggested by the output, the modules ib_ipoib, rdma_ucm and so on are the modules loaded when rdma starts, and are therefore the modules that are needed for this particular HCA. Other HCAs may cause different modules to be loaded.

For a default Red Hat from version 7 onwards, the rdma service can only be started; it cannot be stopped. Finding the modules that load can therefore only be done once for the default configuration, until the next reboot.

The InfiniBand modules that load are the ones that the initrd image needs, so that InfiniBand can be used during the node provisioning stage. The administrator can therefore now create an initrd image with the required InfiniBand modules.

Loading kernel modules into a ramdisk is covered in general in section 5.3.2. A typical Mellanox HCA may have an initrd image created as follows (some text ellipsized in the following example):

Example

```
[root@bright82 ~]# cmsh
[bright82]% softwareimage use default-image
[bright82]->softwareimage[default-image]% kernelmodules
[bright82]->softwareimage[default-image]->kernelmodules]% add mlx4_ib
[bright82]->softwareimage[default-image]->kernelmodules[mlx4_ib]% add ib_ipoib
[bright82]->softwareimage[default-image]->kernelmodules[ib_ipoib]% add ib_umad
[bright82]->softwareimage[default-image]->kernelmodules[ib_umad]% commit
Tue May 24 03:45:35 2011 bright82: Initial ramdisk for image default-image was regenerated successfully.
```

If the modules are put in another image instead of default-image, then the default image that nodes boot from should be set to the new image (section 3.15.2).

InfiniBand Provisioning: Network Configuration

It is assumed that the networking configuration for the final system for InfiniBand is configured following the general guidelines of section 3.6. If it is not, that should be checked first to see if all is well with the InfiniBand network.

The provisioning aspect is set by defining the provisioning interface. An example of how it may be set up for 150 nodes with a working InfiniBand interface ib0 in cmsh is:

Example
5.3 The Kernel Image, Ramdisk And Kernel Modules

5.3.4 Omni-Path Provisioning
The Intel Omni-Path Architecture (OPA) can be set up for provisioning in a similar way to the InfiniBand provisioning described in section 5.3.3.

Omni-Path Provisioning: Ramdisk Image Configuration
The hfi1 and ib_ipoib kernel modules must be added to the software image and the initrd image regenerated:

Example

```
[root@bright82 ~]# cmsh
[bright82]% softwareimage use default-image
[bright82->softwareimage[default-image]]% kernelmodules
[bright82->softwareimage[default-image]->kernelmodules]% add hfi1
[bright82->softwareimage[default-image]->kernelmodules*[hfi1+]]% add ib_ipoib
[bright82->softwareimage[default-image]->kernelmodules*[ib_ipoib+]]% commit
```

Thu Jul 4 15:03:35 2019 bright82: Initial ramdisk for image default-image was regenerated successfully.

Omni-Path Provisioning: Network Configuration
For the Omni-Path interface on the node, the value of provisioninginterface is set to ib0, since it still diverges very little from InfiniBand. So, in same way as for InfiniBand before:

Example

```
[root@bright82 ~]# cmsh
[bright82]% device
[bright82->device]% foreach -n node001..node150 (set provisioninginterface ib0)
```

5.3.5 VLAN Provisioning
Nodes can be configured for provisioning over a VLAN interface, starting in Bright Cluster Manager version 8.2.

This requires:

- A VLAN network and node interface. The VLAN network is typically specified by the network switch. The interface that connects the node to the switch can be configured as a VLAN interface as outlined in section 3.4.

- The BIOS of the node must have the VLANID value set within the BIOS network options. If the BIOS does not support this setting, then PXE over VLAN cannot work. For a NIC that is missing this in the BIOS, the NIC hardware provider may sometimes have a BIOS update that supports this setting.

- The VLANID value should be set in the kernel parameters. Kernel parameters for a node can be specified in cmsh with the kernelparameters setting of softwareimage mode (section 11.3.4). An example where the VLANID is appended to some existing parameters could be:

Example

```
```
5.4 Node-Installer

After the kernel has started up, and the ramdisk kernel modules are in place on the node, the node launches the node-installer.

The node-installer interacts with CMDaemon on the head node and takes care of the rest of the boot process.

As an aside, the node-installer modifies some files (Appendix A.3) on the node it is installing to, so that they differ from the otherwise-expected pre-init stage Linux system. Such modifications can be prevented by a `frozenFilesPerNode` or `frozenFilesPerCategory` directive, as documented within `/cm/node-installer/scripts/node-installer.conf`, and illustrated on page 645.

Once the node-installer has completed its tasks, the local drive of the node has a complete Linux pre-init stage system. The node-installer ends by calling `/sbin/init` from the local drive and the boot process then proceeds as a normal Linux boot.

The steps the node-installer goes through for each node are:

1. requesting a node certificate (section 5.4.1)
2. deciding or selecting node configuration (section 5.4.2)
3. starting up all network interfaces (section 5.4.3)
4. determining install-mode type and execution mode (section 5.4.4)
5. running initialize scripts (section 5.4.5)
6. checking partitions, mounting filesystems (section 5.4.6)
7. synchronizing the local drive with the correct software image (section 5.4.7)
8. writing network configuration files to the local drive (section 5.4.8)
9. creating an `/etc/fstab` file on the local drive (section 5.4.9)
10. installing GRUB bootloader if configured by Bright Cluster Manager (section 5.4.10), and initializing SELinux if it has been installed and configured (Chapter 9 of the Installation Manual)
11. running finalize scripts (section 5.4.11)
12. unloading specific drivers no longer needed (section 5.4.12)
13. switching the root device to the local drive and calling `/sbin/init` (section 5.4.13)

These 13 node-installer steps and related matters are described in detail in the corresponding sections 5.4.1–5.4.13.

5.4.1 Requesting A Node Certificate

Each node communicates with the CMDaemon on the head node using a certificate. If no certificate is found, it automatically requests one from CMDaemon running on the head node (figure 5.7).
The certificate is stored on the head node in /cm/node-installer/certificates/ by MAC address.

**Null Cipher Certificates**

By default, a null cipher is used on internal networks such as internalnet, to keep communications speedy. Using encryption on even these networks is sometimes a requirement in very unusual situations. In that case, setting the advanced configuration flag AllowNullCipherNetwork=0 in cmd.conf (Appendix C, page 657) forces encryption on after CMDaemon is restarted. By default, its value is 1.

**Certificate Auto-signing**

By default, certificate auto-signing means the cluster management daemon automatically issues a certificate to any node that requests a certificate.

For untrusted networks it may be wiser to approve certificate requests manually to prevent new nodes being added automatically without getting noticed. Disabling certificate auto-signing can then be done by issuing the autosign off command from cert mode in cmsh.

Section 2.3 has more information on certificate management in general.

**Example**

Disabling certificate auto-sign mode:

```
[bright82]% cert autosign
on
[bright82]% cert autosign off
off
[bright82]% cert autosign
off
[bright82]%
```

**Certificate Storage And Removal Implications**

After receiving a valid certificate, the node-installer stores it in /cm/node-installer/certificates/<node mac address>/ on the head node. This directory is NFS...
exported to the nodes, but can only be accessed by the root user. The node-installer does not request a new certificate if it finds a certificate in this directory, valid or invalid.

If an invalid certificate is received, the screen displays a communication error. Removing the node’s corresponding certificate directory allows the node-installer to request a new certificate and proceed further.

5.4.2 Deciding Or Selecting Node Configuration

Once communication with the head node CMDaemon is established, the node-installer tries to identify the node it is running on so that it can select a configuration from CMDaemon’s record for it, if any such record exists. It correlates any node configuration the node is expected to have according to network hardware detected. If there are issues during this correlation process then the administrator is prompted to select a node configuration until all nodes finally have a configuration.

Possible Node Configuration Scenarios

The correlations process and corresponding scenarios are now covered in more detail:

It starts with the node-installer sending a query to CMDaemon to check if the MAC address used for net booting the node is already associated with a node in the records of CMDaemon. In particular, it checks the MAC address for a match against the existing node configuration properties, and decides whether the node is known or new.

- the node is known if the query matches a node configuration. It means that node has been booted before.
- the node is new if no configuration is found.

In both cases the node-installer then asks CMDaemon to find out if the node is connected to an Ethernet switch, and if so, to which port. Setting up Ethernet switches for port detection is covered in section 3.8.

If a port is detected for the node, the node-installer queries CMDaemon for a node configuration associated with the detected Ethernet switch port. If a port is not detected for the node, then either the hardware involved with port detection needs checking, or a node configuration must be selected manually.

There are thus several scenarios:

1. The node is new, and an Ethernet switch port is detected. A node configuration associated with the port is found. The node-installer suggests to the administrator that the new node should use this configuration, and displays the configuration along with a confirmation dialog (figure 5.8). This suggestion can be interrupted, and other node configurations can be selected manually instead through a sub-dialog (figure 5.9). By default (in the main dialog), the original suggestion is accepted after a timeout.
2. The node is new, and an Ethernet switch port is detected. A node configuration associated with the port is not found. The node-installer then displays a dialog that allows the administrator to either retry Ethernet switch port detection (figure 5.10) or to drop into a sub-dialog to manually select a node configuration (figure 5.9). By default, port detection is retried after a timeout.
3. The node is new, and an Ethernet switch port is not detected. The node-installer then displays a dialog that allows the user to either retry Ethernet switch port detection (figure 5.11) or to drop into a sub-dialog to manually select a node configuration (figure 5.9). By default, port detection is retried after a timeout.

4. The node is known, and an Ethernet switch port is detected. The configuration associated with the
5.4 Node-Installer

5. The node is known, and an Ethernet switch port is detected. However, the configuration associated with the port is not the same as the configuration associated with the node’s MAC address. This is called a **port mismatch**. This type of port mismatch situation occurs typically during a mistaken node swap, when two nodes are taken out of the cluster and returned, but their positions are swapped by mistake (or equivalently, they are returned to the correct place in the cluster, but the switch ports they connect to are swapped by mistake). To prevent configuration mistakes, the node-installer displays a port mismatch dialog (figure 5.12) allowing the user to retry, accept a node configuration that is associated with the detected Ethernet port, or to manually select another node configuration via a sub-dialog (figure 5.9). By default (in the main port mismatch dialog), port detection is retried after a timeout.

### Figure 5.12: Scenarios: Port Mismatch Dialog

<table>
<thead>
<tr>
<th>Cluster Manager Node Installer</th>
<th>5.0.6096.324</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Port Mismatch Dialog" /></td>
<td></td>
</tr>
</tbody>
</table>

6. The node is known, and an Ethernet switch port is not detected. However, the configuration associated with the node’s MAC address does have an Ethernet port associated with it. This is also considered a port mismatch. To prevent configuration mistakes, the node-installer displays a port mismatch dialog similar to figure 5.12, allowing the user to retry or to drop into a sub-dialog and manually select a node configuration that may work.

However, a more likely solution in most cases is to:

- either clear the switch port configuration in the cluster manager so that switch port detection is not attempted. For example, for node001, this can be done by running this `cmsh` command on the head node:
  ```
  cmsh -c "device clear node001 ethernetswitch"
  ```
- or enable switch port detection on the switch. This is usually quite straightforward, but may
require going through the manuals or software application that the switch manufacturer has provided.

By default (in the port mismatch dialog), port detection is retried after a timeout. This means that if the administrator clears the switch port configuration or enables switch port detection, the node-installer is able to continue automatically with a consistent configuration.

7. The node is known, and an Ethernet switch port is detected. However, the configuration associated with the node’s MAC address has no Ethernet switch port associated with it. This is not considered a port mismatch but an unset switch port configuration, and it typically occurs if switch port configuration has not been carried out, whether by mistake or deliberately. The node-installer displays the configuration as a suggestion along with a confirmation dialog (figure 5.13). The suggestion can be interrupted, and other node configurations can be selected manually instead using a sub-dialog. By default (in the main dialog) the configuration is accepted after a timeout.

<table>
<thead>
<tr>
<th>Scenario</th>
<th>Node known?</th>
<th>Switch port detected?</th>
<th>Switch port configuration found?</th>
<th>Switch port configuration conflicts with node configuration?</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>No</td>
<td>Yes</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>2</td>
<td>No</td>
<td>Yes</td>
<td>No</td>
<td>No</td>
</tr>
<tr>
<td>3</td>
<td>No</td>
<td>No</td>
<td>No</td>
<td>No</td>
</tr>
<tr>
<td>4</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>5</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes (configurations differ)</td>
</tr>
<tr>
<td>6</td>
<td>Yes</td>
<td>No</td>
<td>Yes</td>
<td>Yes (port expected by MAC configuration not found)</td>
</tr>
<tr>
<td>7</td>
<td>Yes</td>
<td>Yes</td>
<td>No</td>
<td>No (port not expected by MAC configuration)</td>
</tr>
</tbody>
</table>
In these scenarios, whenever the user manually selects a node configuration in the prompt dialog, an attempt to detect an Ethernet switch port is repeated. If a port mismatch still occurs, it is handled by the system as if the user has not made a selection.

**Summary Of Behavior During Hardware Changes**

The logic of the scenarios means that an unpreconfigured node always boots to a dialog loop requiring manual intervention during a first install (scenarios 2 and 3). For subsequent boots the behavior is:

- If the node MAC hardware has changed (scenarios 1, 2, 3):
  - if the node is new and the detected port has a configuration, the node automatically boots to that configuration (scenario 1).
  - else manual intervention is needed (scenarios 2, 3)
- If the node MAC hardware has not changed (scenarios 4, 5, 6, 7):
  - if there is no port mismatch, the node automatically boots to its last configuration (scenarios 4, 7).
  - else manual intervention is needed (scenarios 5, 6).

**The newnodes Command**

*newnodes basic use:* New nodes that have not been configured yet can be detected using the `newnodes` command from within the `device` mode of cmsh. A new node is detected when it reaches the node-installer stage after booting, and contacts the head node.

**Example**

```
[bright82->device]% newnodes
The following nodes (in order of appearance) are waiting to be assigned:
MAC First appeared Detected on switch port
------------------------------ ----------------------------- -----------------------
00:0C:29:01:0F:F8 Mon, 14 Feb 2011 10:16:00 CET [no port detected]
```

At this point the node-installer is seen by the administrator to be looping, waiting for input on what node name is to be assigned to the new node.

The nodes can be uniquely identified by their MAC address or switch port address. The port and switch to which a particular MAC address is connected can be discovered by using the `showport` command (section 3.8.4). After confirming that they are appropriate, the `ethernetswitch` property for the specified device can be set to the port and switch values.

**Example**

```
[bright82->device]% showport 00:0C:29:01:0F:F8
switch01:8
[bright82->device]% set node003 ethernetswitch switch01:8
```

When the node name (`node003` in the preceding example) is assigned, the node-installer stops looping and goes ahead with the installation to the node.

The preceding basic use of `newnodes` is useful for small numbers of nodes. For larger number of nodes, the advanced options of `newnodes` may help carry out node-to-MAC assignment with less effort.
newnodes advanced use—options: The list of MAC addresses discovered by a newnodes command can be assigned in various ways to nodes specified by the administrator. Node objects should be created in advance to allow the assignment to take place. The easiest way to set up node objects is to use the --clone option of the foreach command (section 2.5.5).

The advanced options of newnodes are particularly useful for quickly assigning node names to specific physical nodes. All that is needed is to power the nodes up in the right order. For nodes with the same hardware, the node that is powered up first reaches the stage where it tries to connect with the node-installer first. So its MAC address is detected first, and arrives on the list generated by newnodes first. If some time after the first node is powered up, the second node is powered up, then its MAC address becomes the second MAC address on the list, and so on for the third, fourth, and further nodes.

When assigning node names to a physical node, on a cluster that has no such assignment already, the first node that arrived on the list gets assigned the name node001, the second node that arrived on the list gets assigned the name node002 and so on.

The advanced options are shown in device mode by running the help newnodes command. The options can be introduced as being of three kinds: straightforward, grouping, and miscellaneous:

- The straightforward options:
  -n|--nodes
  -w|--write
  -s|--save

  Usually the most straightforward way to assign the nodes is to use the -n option, which accepts a list of nodes, together with a -w or -s option. The -w (--write) option sets the order of nodes to the corresponding order of listed MAC addresses, and is the same as setting an object in cmsh. The -s (--save) option is the same as setting and committing an object in cmsh, so -s implies a -w option is run at the same time.

  So, for example, if 8 new nodes are discovered by the node-installer on a cluster with no nodes so far, then:

  Example

  [bright82->device]% newnodes -w -n node001..node008

  assigns (but does not commit) the sequence node001 to node008 the new MAC address according to the sequence of MAC addresses displaying on the list.

- The grouping options:
  -g|--group
  -c|--category
  -h|--chassis
  -r|--rack

  The “help newnodes” command in device mode shows assignment options other than -n for a node range are possible. For example, the assignments can also be made for a group (-g), per category (-c), per chassis (-h), and per rack (-r).

- The miscellaneous options:
  -f|--force
  -o|--offset

© Bright Computing, Inc.
By default, the `newnodes` command fails when it attempts to set a node name that is already taken. The `-f` (`--force`) option forces the new MAC address to be associated with the old node name. When used with an assignment grouping, (node range, group, category, chassis, or rack) all the nodes in the grouping lose their node-to-MAC assignments and get new assignments. The `-f` option should therefore be used with care.

The `-o` (`--offset`) option takes a number `<number>` and skips `<number>` nodes in the list of detected unknown nodes, before setting or saving values from the assignment grouping.

Examples of how to use the advanced options follow.

**newnodes advanced use---range assignment behavior example:** For example, supposing there is a cluster with nodes assigned all the way up to node022. That is, CMDaemon knows what node is assigned to what MAC address. For the discussion that follows, the three nodes node020, node021, node022 can be imagined as being physically in a rack of their own. This is simply to help to visualize a layout in the discussion and tables that follow and has no other significance. An additional 3 new, that is unassigned, nodes are placed in the rack, and allowed to boot and get to the node-installer stage.

The `newnodes` command discovers the new MAC addresses of the new nodes when they reach their node-installer stage, as before (the switch port column is omitted in the following text for convenience):

```
Example

[bright82->device]# newnodes

<table>
<thead>
<tr>
<th>MAC</th>
<th>First appeared</th>
</tr>
</thead>
<tbody>
<tr>
<td>00:00:29:EF:40:2A</td>
<td>Tue, 01 Nov 2011 11:42:31 CET</td>
</tr>
<tr>
<td>00:00:29:95:D3:5B</td>
<td>Tue, 01 Nov 2011 11:46:25 CET</td>
</tr>
<tr>
<td>00:00:29:65:9A:3C</td>
<td>Tue, 01 Nov 2011 11:47:13 CET</td>
</tr>
</tbody>
</table>
```

The assignment of MAC to node address could be carried out as follows:

```
Example

[bright82->device]# newnodes -s -n node023..node025

<table>
<thead>
<tr>
<th>MAC</th>
<th>First appeared</th>
<th>Hostname</th>
</tr>
</thead>
<tbody>
<tr>
<td>00:00:29:EF:40:2A</td>
<td>Tue, 01 Nov 2011 11:42:31 CET</td>
<td>node023</td>
</tr>
<tr>
<td>00:00:29:95:D3:5B</td>
<td>Tue, 01 Nov 2011 11:46:25 CET</td>
<td>node024</td>
</tr>
<tr>
<td>00:00:29:65:9A:3C</td>
<td>Tue, 01 Nov 2011 11:47:13 CET</td>
<td>node025</td>
</tr>
</tbody>
</table>
```

Once this is done, the node-installer is able to stop looping, and to go ahead and install the new nodes with an image.

The physical layout in the rack may then look as indicated by this:

```
<table>
<thead>
<tr>
<th>before</th>
<th>after</th>
</tr>
</thead>
<tbody>
<tr>
<td>node020</td>
<td>node020</td>
</tr>
<tr>
<td>node021</td>
<td>node021</td>
</tr>
<tr>
<td>node022</td>
<td>node022</td>
</tr>
<tr>
<td></td>
<td>node023</td>
</tr>
<tr>
<td></td>
<td>node024</td>
</tr>
<tr>
<td></td>
<td>node025</td>
</tr>
</tbody>
</table>
```

Here, `node023` is the node with the MAC address ending in A. If instead of the previous `newnodes` command, an offset of 1 is used to skip assigning the first new node:

© Bright Computing, Inc.
Example

[bright82->device]% newnodes -s -o 1 node024..node025

then the rack layout looks like:

<table>
<thead>
<tr>
<th>before</th>
<th>after</th>
<th>MAC</th>
</tr>
</thead>
<tbody>
<tr>
<td>node020</td>
<td>node020</td>
<td></td>
</tr>
<tr>
<td>node021</td>
<td>node021</td>
<td></td>
</tr>
<tr>
<td>node022</td>
<td>node022</td>
<td></td>
</tr>
<tr>
<td>unassigned</td>
<td>...A</td>
<td></td>
</tr>
<tr>
<td>node024</td>
<td>...B</td>
<td></td>
</tr>
<tr>
<td>node025</td>
<td>...C</td>
<td></td>
</tr>
</tbody>
</table>

Here, unassigned is where node023 of the previous example is physically located, that is, the node with the MAC address ...A. The lack of assignment means there is actually no association of the name node023 with that MAC address, due to the newnodes command having skipped over it with the -o option.

If instead the assignment is done with:

Example

[bright82->device]% newnodes -s 1 node024..node026

then the node023 name is unassigned, and the name node024 is assigned instead to the node with the MAC address ...A, so that the rack layout looks like:

<table>
<thead>
<tr>
<th>before</th>
<th>after</th>
<th>MAC</th>
</tr>
</thead>
<tbody>
<tr>
<td>node020</td>
<td>node020</td>
<td></td>
</tr>
<tr>
<td>node021</td>
<td>node021</td>
<td></td>
</tr>
<tr>
<td>node022</td>
<td>node022</td>
<td></td>
</tr>
<tr>
<td>node024</td>
<td>...A</td>
<td></td>
</tr>
<tr>
<td>node025</td>
<td>...B</td>
<td></td>
</tr>
<tr>
<td>node026</td>
<td>...C</td>
<td></td>
</tr>
</tbody>
</table>

newnodes advanced use—assignment grouping example: Node range assignments are one way of using newnodes. However assignments can also be made to a category, a rack, or a chassis. For example, with Bright View assigning node names to a rack can be done from the Racks option of the node. For example, to add a node001 to a rack1, the clickpath would be:
Devices→Settings [node001]→Rack [rack1].

In cmsh, the assignment of multiple node names to a rack can conveniently be done with a foreach loop from within device mode:

Example

[bright82->device]% foreach -n node020..node029 (set rack rack02)
[bright82->device*]% commit
[bright82->device]% foreach -n node030..node039 (set rack rack03)
[bright82->device*]% commit

© Bright Computing, Inc.
The assignment of node names with the physical node in the rack can then be arranged as follows: If
the nodes are identical hardware, and are powered up in numerical sequence, from node020 to node039,
with a few seconds in between, then the list that the basic `newnodes` command (without options) displays
is arranged in the same numerical sequence. Assigning the list in the rack order can then be done by
running:

**Example**

```bash
[bright82->device] newnodes -s -r rack02..rack03
```

If it turns out that the boot order was done very randomly and incorrectly for all of rack02, and that
the assignment for rack02 needs to be done again, then a simple way to deal with it is to bring down the
nodes of rack02, then clear out all of the rack02 current MAC associations, and redo them according to
the correct boot order:

**Example**

```bash
[bright82->device] foreach -r rack02 ( clear mac ) ; commit

[...removes MAC association with nodes from CMDaemon...]

[...now reboot nodes in rack02 in sequence (not with Bright Cluster Manager)...]

[bright82->device] newnodes

[...shows sequence as the nodes come up...]

[bright82->device] newnodes -s -r rack02

[...assigns sequence in boot order...]
```

**newnodes advanced use—assignment forcing example:** The `--force` option can be used in the follow-
ing case: Supposing that node022 fails, and a new node hardware comes in to replace it. The new regular
node has a new MAC address. So, as explained by scenario 3 (section 5.4.2), if there is no switch port
assignment in operation for the nodes, then the node-installer loops around, waiting for intervention.\(^1\)

This situation can be dealt with from the command line by:

- accepting the node configuration at the regular node console, via a sub-dialog
- accepting the node configuration via `cmsh`, without needing to be at the regular node console:

```bash
[bright82->device] newnodes -s -f -n node022
```

**Node Identification**

The node identification resource can be accessed via the clickpath:

`Devices` → `Nodes Identification`

The node identification resource is roughly the Bright View equivalent to the `newnodes` command of
`cmsh`. Like `newnodes`, the resource lists the MAC address of any unassigned node that the head node
detects, and shows the associated detected switch port for the node. Also, like `newnodes`, it can help
assign a node name to the node, assuming the node object exists. After assignment is done, the new
status should be saved.

\(^1^\)with switch port assignment in place, scenario 1 means the new node simply boots up by default and becomes the new
node022 without further intervention

© Bright Computing, Inc.
The most useful way of using the node identification resource is for node assignment in large clusters.

To do this, it is assumed that the node objects have already been created for the new nodes. The creation of the node objects means that the node names exist, and so assignment to the node names is able to take place. An easy way to create nodes, set their provisioning interface, and set their IP addresses is described in the section on the node creation wizard (section 5.7.2). Node objects can also be created by running cmsh’s foreach loop command on a node with a --clone option (section 2.5.5).

The nodes are also assumed to be set for net booting, typically set from a BIOS setting.

The physical nodes are then powered up in an arranged order. Because they are unknown new nodes, the node-installer keeps looping after a timeout. The head node in the meantime detects the new MAC addresses and switch ports in the sequence in which they first have come up and lists them in that order.

By default, all these newly detected nodes are set to auto, which means their numbering goes up sequentially from whatever number is assigned to the preceding node in the list. Thus, if there are 10 new unassigned nodes that are brought into the cluster, and the first node in the list is assigned to the first available number, say node327; then clicking on assign automatically assigns the remaining nodes to the next available numbers, say node328–node337.

After the assignment, the node-installer looping process on the new nodes notices that the nodes are now known. The node-installer then breaks out of the loop, and installation goes ahead without any intervention needed at the node console.

5.4.3 Starting Up All Network Interfaces

At the end of section 5.4.2, the node-installer knows which node it is running on, and has decided what its node configuration is.

Starting Up All Provisioning Network Interfaces

It now gets on with setting up the IP addresses on the provisioning interfaces required for the node-installer, while taking care of matters that come up on the way:

Avoiding duplicate IP addresses: The node-installer brings up all the network interfaces configured for the node. Before starting each interface, the node-installer first checks if the IP address that is about to be used is not already in use by another device. If it is, then a warning and retry dialog is displayed until the IP address conflict is resolved.

Using BOOTIF to specify the boot interface: BOOTIF is a special name for one of the possible interfaces. The node-installer automatically translates BOOTIF into the name of the device, such as eth0 or eth1, used for network booting. This is useful for a machine with multiple network interfaces where it can be
unclear whether to specify, for example, `eth0` or `eth1` for the interface that was used for booting. Using the name `BOOTIF` instead means that the underlying device, `eth0` or `eth1` in this example, does not need to be specified in the first place.

**Halting on missing kernel modules for the interface:** For some interface types like VLAN and channel bonding, the node-installer halts if the required kernel modules are not loaded or are loaded with the wrong module options. In this case the kernel modules configuration for the relevant software image should be reviewed. Recreating the ramdisk and rebooting the node to get the interfaces up again may be necessary, as described in section 5.8.5.

**Bringing Up Non-Provisioning Network Interfaces**
Provisioning interfaces are by default automatically brought up during the init stage, as the node is fully booted up. The BMC and non-provisioning interfaces on the other hand have a different behavior:

**Bringing Up And Initializing BMC Interfaces:** If a BMC interface is present and powered up, then it is expected to be running at least with layer 2 activity (ethernet). It can be initialized in the node configuration (section 3.7) with an IP address, netmask and user/password settings so that layer 3 (TCP/IP) networking works for it. BMC networking runs independently of node networking.

**Bringing up non-BMC, non-provisioning network interfaces:** Non-provisioning interfaces are inactive unless they are explicitly brought up. Bright Cluster Manager can configure how these non-provisioning interfaces are brought up by using the `bringupduringinstall` parameter, which can take the following values:

- **yes**: Brings the interface up during the pre-init stage
- **no**: Keeps the interface down during the pre-init stage. This is the default for non-provisioning interfaces.
- **yesandkeep**: Brings the interface up during the pre-init stage, and keeps it up during the transition to the init stage.

**Bringing Up And Keeping Up Provisioning Network Interfaces**
The preceding `bringupduringinstall` parameter is not generally supported for provisioning interfaces. However the `yesandkeep` value does work for provisioning interfaces too, under some conditions:

- **yesandkeep**: Brings the interface up during the pre-init stage, and keeps it up during the transition to the init stage, for the following provisioning devices:
  - Ethernet device interfaces using a leased DHCP address
  - InfiniBand device interfaces running with distribution OFED stacks

**Restarting The Network Interfaces**
At the end of this step (i.e. section 5.4.3) the network interfaces are up. When the node-installer has completed the remainder of its 13 steps (sections 5.4.4–5.4.13), control is handed over to the local `init` process running on the local drive. During this handover, the node-installer brings down all network devices. These are then brought back up again by `init` by the distribution's standard networking `init` scripts, which run from the local drive and expect networking devices to be down to begin with.
5.4.4 Determining Install-mode Type And Execution Mode

Stored install-mode values decide whether synchronization is to be applied fully to the local drive of the node, only for some parts of its filesystem, not at all, or even whether to drop into a maintenance mode instead.

Related to install-mode values are execution mode values (page 179) that determine whether to apply the install-mode values to the next boot, to new nodes only, to individual nodes or to a category of nodes.

Related to execution mode values is the confirmation requirement toggle value (page 181) in case a full installation is to take place.

These values are merely determined at this stage; nothing is executed yet.

Install-mode Values

The install-mode can have one of four values: AUTO, FULL, MAIN and NOSYNC. It should be understood that the term “install-mode” implies that these values operate only during the node-installer phase.²

- If the install-mode is set to FULL, the node-installer re-partitions, creates new filesystems and synchronizes a full image onto the local drive according a partition layout. This process wipes out all pre-boot drive content.

A partition layout (Appendix D) includes defined values for the partitions, sizes, and filesystem types for the nodes being installed. An example of a partition layout is the default partition layout (Appendix D.3).

- If the install-mode is set to AUTO, the node-installer checks the partition layout of the local drive against the node’s stored configuration. If these do not match because, for example, the node is new, or if they are corrupted, then the node-installer recreates the partitions and filesystems by carrying out a FULL install. If however the drive partitions and filesystems are healthy, the node-installer only does an incremental software image synchronization. Synchronization tends to be quick because the software image and the local drive usually do not differ much.

Synchronization also removes any extra local files that do not exist on the image, for the files and directories considered. Section 5.4.7 gives details on how it is decided what files and directories are considered.

- If the install-mode is set to MAIN, the node-installer halts in maintenance mode, allowing manual investigation of specific problems. The local drive is untouched.

- If the install-mode is set to NOSYNC, and the partition layout check matches the stored XML configuration, then the node-installer skips synchronizing the image to the node, so that contents on the local drive persist from the previous boot. An exception to this is the node certificate and key, that is the files /cm/local/apps/cmd/etc/cert.{pem|key}. These are updated from the head node if missing.

If however the partition layout does not match the stored configuration, a FULL image sync is triggered. Thus, for example, a burn session (Chapter 8 of the Installation Manual), with the default burn configuration which destroys the existing partition layout on a node, will trigger a FULL image sync on reboot after the burn session.

The NOSYNC setting should therefore not be regarded as a way to protect data. Ways to preserve data across node reboots are discussed in the section that discusses the FULL install confirmation settings (page 181).

NOSYNC is useful during mass planned node reboots when set with the nextinstallmode option of device mode. This sets the nodes to use the OS on the hard drive, during the next boot only, without an image sync;

²For example, imageupdate (section 5.6.2), which is run by CMDaemon, ignores these settings, which is as expected. This means that, for example, if imageupdate is run with NOSYNC set, then the head node image is still synchronized over as usual to the regular node while the node is up. It is only during node boot, during the installer stage, that setting NOSYNC prevents synchronization.
**Example**

```bash
[bright82]% device foreach -n node001..node999 (set nextinstallmode nosync)
[bright82]% device commit
```

**Install-mode Logging**

The decision that is made is normally logged to the node-installer file, `/var/log/node-installer` on the head node.

**Example**

```
08:40:58 node001 node-installer: Installmode is: AUTO
08:40:58 node001 node-installer: Fetching disks setup.
08:40:58 node001 node-installer: Setting up environment for initialize scripts.
08:40:58 node001 node-installer: Initialize script for category default is empty.
08:40:59 node001 node-installer: Checking partitions and filesystems.
08:40:59 node001 node-installer: Updating device status: checking disks
08:40:59 node001 node-installer: Detecting device '/dev/sda': found
08:41:00 node001 node-installer: Number of partitions on sda is ok.
08:41:00 node001 node-installer: Size for /dev/sda1 is ok.
08:41:00 node001 node-installer: Checking if /dev/sda1 contains ext3 filesystem.
08:41:01 node001 node-installer: fsck.ext3 --a /dev/sda1
08:41:01 node001 node-installer: /dev/sda1: recovering journal
08:41:02 node001 node-installer: Number of partitions on sda is ok.
08:41:02 node001 node-installer: Size for /dev/sda1 is ok.
08:41:02 node001 node-installer: Checking partitions and filesystems.
08:41:02 node001 node-installer: Filesystem check on /dev/sda1 is ok.
08:41:02 node001 node-installer: Size for /dev/sda2 is wrong.
08:41:02 node001 node-installer: Partitions and/or filesystems are missing/corrupt. (Exit code 18, signal 0)
08:41:03 node001 node-installer: Creating new disk layout.
```

In this case the node-installer detects that the size of `/dev/sda2` on the disk no longer matches the stored configuration, and triggers a full re-install. For further detail beyond that given by the node-installer log, the `disks` script at `/cm/node-installer/scripts/disks` on the head node can be examined. The node-installer checks the disk by calling the `disks` script. Exit codes, such as the 18 reported in the log example, are defined near the top of the `disks` script.

**Install-mode’s Execution Modes**

Execution of an install-mode setting is possible in several ways, both permanently or just temporarily for the next boot. Execution can be set to apply to categories or individual nodes. The node-installer looks for install-mode execution settings in this order:

1. The “New node installmode” property of the node’s category. This decides the install mode for a node that is detected to be new.
   - It can be set for the default category using a clickpath such as:
     ```
     Grouping→Node categories[default]→Edit→Settings→Install mode
     ```
   - or using `cmsh` with a one-liner such as:
     ```cmsh -c "category use default; set newnodeinstallmode FULL; commit"
     ```
   - By default, the “New node installmode” property is set to FULL.

2. The Install-mode setting as set by choosing a PXE menu option on the console of the node before it loads the kernel and ramdisk (figure 5.15). This only affects the current boot. By default the PXE menu install mode option is set to AUTO.
3. The "Next boot install-mode" property of the node configuration. This can be set for a node such as node001 using a clickpath such as:

Devices → Nodes[node001] → Edit → Settings → Install mode

It can also be set using cmsh with a one-liner like:

```
cmsh -c "device use node001; set nextinstallmode FULL; commit"
```

The property is cleared when the node starts up again, after the node-installer finishes its installation tasks. So it is empty unless specifically set by the administrator during the current uptime for the node.

4. The install-mode property can be set in the node configuration using Bright View via

Devices → Nodes[node001] → Edit → Settings → Install mode or using cmsh with a one-liner such as:

```
cmsh -c "device use node001; set installmode FULL; commit"
```

By default, the install-mode property is auto-linked to the property set for install-mode for that category of node. Since the property for that node’s category defaults to AUTO, the property for the install-mode of the node configuration defaults to “AUTO (Category)”.

5. The install-mode property of the node’s category. This can be set using Bright View with a clickpath such as:
5.4 Node-Installer

Grouping → Node categories [default] → Edit → Settings → Install mode
or using cmsh with a one-liner such as:

```bash
cmsh -c "category use default; set installmode FULL; commit"
```

As already mentioned in a previous point, the install-mode is set by default to AUTO.

6. A dialog on the console of the node (figure 5.16) gives the user a last opportunity to overrule the install-mode value as determined by the node-installer. By default, it is set to AUTO:

![Figure 5.16: Install-mode Setting Option During Node-Installer Run](image)

**FULL Install Confirmation via datanode Setting**

Related to execution mode values is the ability to carry out a FULL install only after explicit confirmation, via the datanode property. This must be set in order to prompt for a confirmation, when a FULL installation is about to take place. If it is set, then the node-installer only goes ahead with the FULL install after the administrator has explicitly confirmed it.

The datanode property can be set in the node configuration of, for example, node001 with Bright View via the clickpath:

Devices → Nodes [node001] → Edit → Settings → Data node [Yes]

Alternatively, the parameter datanode can be set using a cmsh one-liner as follows:

```bash
[root@bright82 ~]# cmsh -c "device use node001; set datanode yes; commit"
```

The property can also be set at a category or other levels.

**Why the FULL install confirmation is useful:** The reason for such a setting is that a FULL installation can be triggered by disk or partition changes, or by a change in the MAC address. If that happens, then:

- considering a drive, say, `/dev/sda` that fails, this means that any drive `/dev/sdb` would normally become `/dev/sda` upon reboot. In that case an unwanted FULL install would not only be triggered by an install-mode settings of FULL, but also by the install-mode settings of AUTO or
NOSYNC. Having the new, “accidental” /dev/sda have a FULL install is unlikely to be the intention, since it would probably contain useful data that the node-installer earlier left untouched.

- considering a node with a new MAC address, but with local storage containing useful data from earlier. In this case, too, an unwanted FULL install would not only be triggered by an install-mode setting of FULL, but also by the install-mode settings AUTO or NOSYNC.

Thus, in cases where nodes are used to store data, an explicit confirmation before overwriting local storage contents is a good idea. However, by default, no confirmation is asked for when a FULL installation is about to take place.

**Carrying out the confirmation:** When the confirmation is required, then it can be carried out by the administrator as follows:

- From the node console. A remote console launched from Bright View or cmsh will also work if SOL connectivity has been configured.
- From cmsh, within device mode, using the installerinteractions command (some output elided):

  **Example**

  ```
  [bright82->device]% installerinteractions -w -n node001 --confirm
  Hostname   Action
  ---------- ---------------------------------------------
  node001    Requesting FULL Install (partition mismatch)
  [bright82->device]%
  ...
  [notice] bright82: node001 [ INSTALLER_CALLINGINIT ]...
  [bright82->device]%
  ...
  [notice] bright82: node001 [ UP ]
  ```

  Besides confirmation, the installerinteractions command has options that include letting it:

  - deny the installation, and put it into maintenance mode
  - carry out a dry-run
  - carry out its actions for node groupings such as: node lists, node categories, node groups, chassis, racks, as are possible in the grouping options (page 37).

  Further details on the command can be viewed by running help installerinteractions.

  **An alternative way to avoid overwriting node storage:** Besides the method of FULL install confirmation for datanode, there is a method based on XML assertions, that can also be used to prevent data loss on nodes.

  It uses XML assertions to confirm that the physical drive is recognized (Appendix D.11).

  **A way to overwrite a specified block device:** A related method is that sometimes, for reasons of performance or convenience, it may be desirable to clear data on particular block devices for a node, and carry it out during the next boot only. This can done by setting the block device names to be cleared as values to the parameter Block devices cleared on next boot. The values can be set in cmsh as follows:

  ```
  [bright82->device[node001]]% append blockdevicesclearedonnextboot /dev/sda /dev/sdb ; commit
  ```

  The value of blockdevicesclearedonnextboot is automatically cleared after the node is rebooted.
5.4 Node-Installer

5.4.5 Running Initialize Scripts

An initialize script is used when custom commands need to be executed before checking partitions and mounting devices (section 3.15.4). For example, to initialize some not explicitly supported hardware, or to do a RAID configuration lookup for a particular node. In such cases the custom commands are added to an initialize script. How to edit an initialize script is described in Appendix E.2.

An initialize script can be added to both a node’s category and the node configuration. The node-installer first runs an initialize script, if it exists, from the node’s category, and then an initialize script, if it exists, from the node’s configuration.

The node-installer sets several environment variables which can be used by the initialize script. Appendix E contains an example script documenting these variables.

Related to the initialize script is the finalize script (section 5.4.11). This may run after node provisioning is done, but just before the init process on the node runs.

5.4.6 Checking Partitions, RAID Configuration, Mounting Filesystems

Behavior As Decided By The Install-Mode Value

In section 5.4.4 the node-installer determines the install-mode value, along with when to apply it to a node.

**AUTO:** The install-mode value is typically set to default to AUTO. If AUTO applies to the current node, it means the node-installer then checks the partitions of the local drive and its filesystems and recreates them in case of errors. Partitions are checked by comparing the partition layout of the local drive(s) against the drive layout as configured in the node’s category configuration and the node configuration.

After the node-installer checks the drive(s) and, if required, recreates the layout, it mounts all filesystems to allow the drive contents to be synchronized with the contents of the software image.

**FULL or MAIN:** If install-mode values of FULL or MAIN apply to the current node instead, then no partition checking or filesystem checking is done by the node-installer.

**NOSYNC:** If the install-mode value of NOSYNC applies, then if the partition and filesystem checks both show no errors, the node starts up without getting an image synced to it from the provisioning node. If the partition or the filesystem check show errors, then the node partition is rewritten, and a known good image is synced across.

Behavior As Decided By XML Configuration Settings

The node-installer is capable of creating advanced drive layouts, including LVM setups, and hardware and software RAID setups. Drive layout examples and relevant documentation are in Appendix D.

The XML description used to set the drive layouts can be deployed for a single device or to a category of devices.

**Hardware RAID:** Bright Cluster Manager supports hardware RAID levels 0, 1, 5, 10, and 50, and supports the following options:

<table>
<thead>
<tr>
<th>Option</th>
</tr>
</thead>
<tbody>
<tr>
<td>64kB</td>
</tr>
<tr>
<td>128kB</td>
</tr>
<tr>
<td>256kB</td>
</tr>
<tr>
<td>512kB</td>
</tr>
<tr>
<td>1024kB</td>
</tr>
</tbody>
</table>

© Bright Computing, Inc.
184 Node Provisioning

- **cache policy:**
  - Cached
  - Direct

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>NORA</td>
<td>No Read Ahead</td>
</tr>
<tr>
<td>RA</td>
<td>Read Ahead</td>
</tr>
<tr>
<td>ADRA</td>
<td>Adaptive Read</td>
</tr>
</tbody>
</table>

- **write policy:**
  - WT Write Through
  - WB Write Back

5.4.7 Synchronizing The Local Drive With The Software Image

After having mounted the local filesystems, these can be synchronized with the contents of the software image associated with the node (through its category). Synchronization is skipped if NOSYNC is set, and takes place if install-mode values of FULL or AUTO are set. Synchronization is delegated by the node-installer to the CMDaemon provisioning system. The node-installer just sends a provisioning request to CMDaemon on the head node.

For an install-mode of FULL, or for an install-mode of AUTO where the local filesystem is detected as being corrupted, full provisioning is done. For an install-mode of AUTO where the local filesystem is healthy and agrees with that of the software image, sync provisioning is done.

The software image that is requested is available to nodes by default. It can however be set to a locked state, which means that the request is held in the queue until it is no longer in a locked state. This is sometimes useful for making changes to an image when nodes are booting.

**Example**

```
[root@bright82 ~]# cmsh
[bright82]% softwareimage use default-image
[bright82->softwareimage/default-image]% set locked yes
[bright82->softwareimage/default-image]% commit
```

For an unlocked image, on receiving the provisioning request, CMDaemon assigns the provisioning task to one of the provisioning nodes. The node-installer is notified when image synchronization starts, and also when the image synchronization task ends—whether it is completed successfully or not.

**Exclude Lists:** excludelistsyncinstall And excludelistfullinstall

What files are synchronized is decided by an exclude list. An exclude list is a property of the node category, and is a list of directories and files that are excluded from consideration during synchronization. The excluded list that is used is decided by the type of synchronization chosen: full or sync:

- A full type of synchronization rewrites the partition table of the node, then copies the filesystem from a software image to the node, using a list to specify files and directories to exclude from consideration when copying over the filesystem. The list of exclusions used is specified by the excludelistfullinstall property.

The intention of full synchronization is to allow a complete working filesystem to be copied over from a known good software image to the node. By default the excludelistfullinstall list contains /proc/, /sys/, and lost+found/, which have no content in Bright Cluster Manager’s default
software image. The list can be modified to suit the requirements of a cluster, but it is recommended to have the list adhere to the principle of allowing a complete working node filesystem to be copied over from a known good software image.

- A sync type of synchronization uses the property `excludelistsyncinstall` to specify what files and directories to exclude from consideration when copying parts of the filesystem from a known good software image to the node. The `excludelistsyncinstall` property is in the form of a list of exclusions, or more accurately in the form of two sub-lists.

  The contents of the sub-lists specify the parts of the filesystem that should be retained or not copied over from the software image during sync synchronization when the node is booting. The intention behind this is to have the node boot up quickly, updating only the files from the image to the node that need updating due to the reboot of the node, and otherwise keeping files that are already on the node hard disk unchanged. The contents of the sub-lists are thus items such as the node log files, or items such as the `/proc` and `/sys` pseudo-filesystems which are generated during node boot.

  The administrator should be aware that nothing on a node hard drive can be regarded as persistent because a FULL sync takes place if any error is noticed during a partition or filesystem check.

  Anything already on the node that matches the content of these sub-lists is not overwritten by image content during an `excludelistsyncinstall` sync. However, image content that is not on the node is copied over to the node only for items matching the first sub-list. The remaining files and directories on the node, that is, the ones that are not in the sub-lists, lose their original contents, and are copied over from the software image.

  A `cmsh` one-liner to get an exclude list for a category is:

  ```
  cmsh -c "category use default; get excludelistfullinstall"
  ```

  Similarly, to set the list:

  ```
  cmsh -c "category use default; set excludelistfullinstall; commit"
  ```

  where a text-editor opens up to allow changes to be made to the list. In Bright View the clickpath is:

  Grouping→Node Categories→Edit→Node Category→Settings→Exclude list full install

  Image synchronization is done using `rsync`, and the syntax of the items in the exclude lists conforms to the “INCLUDE/EXCLUDE PATTERN RULES” section of the `rsync(1)` man page, which includes patterns such as “**”, “?”, and “[[:alpha:]]”.

  The `excludelistfullinstall` and `excludelistsyncinstall` properties decide how a node synchronizes to an image during boot. For a node that is already fully up, the related `excludelistupdate` property decides how a running node synchronizes to an image without a reboot event, and is discussed in section 5.6.

**Interface Used To Receive Image Data:** `provisioninginterface`

For regular nodes with multiple interfaces, one interface may be faster than the others. If so, it can be convenient to receive the image data via the fastest interface. Setting the value of `provisioninginterface`, which is a property of the node configuration, allows this.

By default it is set to `BOOTIF` for regular nodes. Using `BOOTIF` is not recommended for node configurations with multiple interfaces.

When listing the network interfaces in `cmsh`, the provisioning interface has a [prov] flag appended to its name.

**Example**
Node Provisioning

Head nodes and provisioning interface: A head node in a single-head cluster does not use the provisioning interface setting.

Head nodes in a failover configuration (Chapter 14), however, do have a value set for provisioning interface, corresponding to the interface on the head that is being provisioned over internalnet by the other head (eth0 in figure 14.1).

Transport Protocol Used For Image Data: provisioningtransport

The provisioning transport property of the node sets whether the image data is sent encrypted or unencrypted to the node from the provisioner. The property value is set via the device mode for the receiving node to one of these values:

- rsyncdaemon, which sends the data unencrypted
- rsyncssh, which sends the data encrypted

The provisioning transport value can be set for all nodes, including provisioning nodes, head nodes, and cloud-director (section 3.2 of the Cloudbursting Manual) nodes. Because encryption severely increases the load on the provisioning node, using rsyncssh is only suggested if the users on the network cannot be trusted. By default, provisioningtransport is set to rsyncdaemon. If high availability (chapter 14) is set up with the head nodes exposed to the outside world on the external network, the administrator should consider setting up rsyncssh for the head nodes.

The rsyncssh transport requires passwordless root access via ssh from the provisioner to the node being provisioned. This is configured by default in the default Bright Cluster Manager nodes. However, if a new image is created with the --exclude options for cm-create-image as explained in (section 11.6.2), the keys must be copied over from /root/.ssh/ on the existing nodes.

Tracking The Status Of Image Data Provisioning: provisioningstatus

The provisioningstatus command within the softwareimage mode of cmsh displays an updated state of the provisioning system. As a one-liner, it can be run as:

```
bright82:~ # cmsh -c "softwareimage provisioningstatus"
Provisioning subsystem status: idle, accepting requests
Update of provisioning nodes requested: no
Maximum number of nodes provisioning: 10000
Nodes currently provisioning: 0
Nodes waiting to be provisioned: <none>
Provisioning node bright82:
  Max number of provisioning nodes: 10
  Nodes provisioning: 0
  Nodes currently being provisioned: <none>
```

The provisioningstatus command has several options that allow the requests to be tracked. The -r option displays the basic status information on provisioning requests, while the -a option displays all status information on provisioning requests. Both of these options display the request IDs.

The Bright View equivalent to provisioningstatus is accessed via the clickpath:

Provisioning -> Provisioning nodes

By default, it displays basic status information on provisioning requests.
Tracking The Provisioning Log Changes: `synclog`

For a closer look into the image file changes carried out during provisioning requests, the `synclog` command from `device` mode can be used (lines elided in the following output):

**Example**

    [bright82->device]% synclog node001
    Tue, 11 Jan 2011 13:27:17 CET - Starting rsync daemon based provisioning. Mode is SYNC.

    sending incremental file list
    ./
    ...
    deleting var/lib/ntp/etc/localtime
    var/lib/ntp/var/run/ntp/
    ...
    sent 2258383 bytes received 6989 bytes 156232.55 bytes/sec
    total size is 1797091769 speedup is 793.29


**Aborting Provisioning With `cancelprovisioningrequest`**

The `cancelprovisioningrequest` command cancels provisioning.

Its usage is:

    cancelprovisioningrequest [OPTIONS] [<requestid> ...]

To cancel all provisioning requests, it can be run as:

    bright82:~ # cmsh -c "softwareimage cancelprovisioningrequest -a"

The `provisioningstatus` command of `cmsh`, can be used to find request IDs. Individual request IDs, for example 10 and 13, can then be specified in the `cancelprovisioningrequest` command, as:

    bright82:~ # cmsh -c "softwareimage cancelprovisioningrequest 10 13"

The help page for `cancelprovisioningrequest` shows how to run the command on node ranges, groups, categories, racks, chassis, and so on.

The Bright View equivalents to the `cmsh` versions for managing provisioning requests can be accessed via the clickpath `Provisioning → Provisioning Requests`.

### 5.4.8 Writing Network Configuration Files

In the previous section, the local drive of the node is synchronized according to install-mode settings with the software image from the provisioning node. The node-installer now sets up configuration files for each configured network interface. These are files like:

    /etc/sysconfig/network-scripts/ifcfg-eth0

for Red Hat, Scientific Linux, and CentOS, while SUSE would use:

    /etc/sysconfig/network/ifcfg-eth0

These files are placed on the local drive.

When the node-installer finishes its remaining tasks (sections 5.4.9–5.4.13) it brings down all network devices and hands over control to the local `/sbin/init` process. Eventually a local `init` script uses the network configuration files to bring the interfaces back up.
5.4.9 Creating A Local /etc/fstab File

The /etc/fstab file on the local drive contains local partitions on which filesystems are mounted as the init process runs. The actual drive layout is configured in the category configuration or the node configuration, so the node-installer is able to generate and place a valid local /etc/fstab file. In addition to all the mount points defined in the drive layout, several extra mount points can be added. These extra mount points, such as NFS imports, /proc, /sys and /dev/shm, can be defined and managed in the node’s category and in the specific configuration of the node configuration, using Bright View or cmsh (section 3.10.2).

5.4.10 Installing GRUB Bootloader

By default, a node-installer boots from the software image on the head node via the network. Optionally, the node-installer installs a boot record on the local drive if the installbootrecord property of the node configuration or node category is set to on, so that the next boot can be from the local drive.

For a hard drive boot to work:

1. hard drive booting must be set to have a higher priority than network booting in the BIOS of the node. Otherwise regular PXE booting occurs, despite whatever value installbootrecord has.

2. the GRUB bootloader with a boot record must be installed in the MBR of the local drive, overwriting the default iPXE boot record.

If the administrator is not using the default software image, but is using a custom software image (section 11.6.1), and if the image is based on a running node filesystem that has not been built directly from a parent distribution, then the GRUB boot configuration may not be appropriate for a standalone GRUB boot to work. This is because the parent distribution installers often use special logic for setting up the GRUB boot configuration. Carrying out this same special logic for all distributions using the custom software image creation tool cm-create-image (section 11.6.2) is impractical.

Providing a custom working image from a standalone node that has been customized after direct installation from the parent distribution, ensures the GRUB boot configuration layout of the custom image is as expected by the parent distribution. This then allows a standalone GRUB boot on the node to run properly.

With a working custom software image, to set the GRUB bootloader in Bright View, the Install boot record option must be enabled and saved in the node configuration or in the node category. The cmsh equivalents are commands like:

```cmsh
cmsh -c "device use node001; set installbootrecord yes; commit"
```

or

```cmsh
cmsh -c "category use default; set installbootrecord yes; commit"
```

Arranging for the two items in the preceding list ensures that the next boot is from GRUB on the hard drive. However, the BOOTIF needs to be changed for booting to be successful. How it can be changed, and why it needs to be changed, is described next.

Changing BOOTIF To Ensure The Node Can Be Standalone

If the BIOS is set to boot from the hard drive, and the boot record has been installed as in the previous two items, then the node boots via the boot record on the hard drive.

BOOTIF is the default value for the network interface for a node that is configured as a Bright software image. However, the BOOTIF interface is undefined during hard drive booting, because it depends on the network provisioning setup, which is not running. This means that the networking interface would
fail during hard drive boot for a standard image. To remedy this, the interface should be set to a defined 
network device name, such as eth0, or the modern equivalents such as en01 (section 5.8.1). The defined 
network device name, as the kernel sees it, can be found by logging into the node and taking a look at 
the output of ip link:

Example

[root@bright82 ~]# ssh node001 ip link
1: lo: <LOOPBACK,UP,LOWER_UP> mtu 65536 qdisc noqueue...
2: eth0: <BROADCAST,MULTICAST,UP,LOWER_UP> mtu 1500 qdisc ... 

In Bright Cluster Manager the provisioning interface is mandatory, even if it is not provisioning. So 
it is set to the value of kernel-defined network device name instead of BOOTIF:

Example

[bright82]% device interfaces node001
[bright82->device[node001]->interfaces]% list

Type Network device name IP Network
------------ -------------------- ---------------- ----------------
physical BOOTIF [prov] 10.141.0.1 internalnet

[bright82->device[node001]->interfaces]% set bootif networkdevicename eth0
[bright82->device[node001]->interfaces]% commit
[bright82->device[node001]->interfaces]% list

Type Network device name IP Network
------------ -------------------- ---------------- ----------------
physical eth0 [prov] 10.141.0.1 internalnet

Tue Mar 31 13:46:44 2020 [notice] bright82: node001 [ UP ], restart required (eth0)

In the preceding example, the kernel-defined network device name is assumed to be eth0. It should 
be modified as required.

In addition, the new IP address is assumed to be on the same internal network. If the administrator 
intends the node to be standalone on another network, then the network and the IP address can be set 
to appropriate values.

When interface changes are carried out to make the node standalone, warnings show up saying that 
a reboot is required. A reboot of the node should be done when the interface configuration is complete. 
During reboot, the node then boots from the hard drive as a standalone, with a non-BOOTIF network 
interface.

Bringing A Node That Boots From Its Hard Drive Back Into A Cluster

If the node is to be brought back into the cluster, then simply unsetting “Install boot record” and rebooting the node does not restore its iPXE boot record and hence its ability to iPXE boot. To restore the iPXE boot record, the node can be booted from the default image copy on the head node via a network boot again. Typically this is done by manual intervention during node boot to select network booting from the BIOS of the node.

Setting the value of provisioninginterface in cmsh for the node to BOOTIF is also recommended.

As suggested by the Bright Cluster Manager iPXE boot prompt, setting network booting to work from the BIOS (regular “PXE” booting) is preferred to the relatively roundabout way of iPXE booting from the disk.

SELinux Initialization For Hard Drive Boot And PXE Boot

If configured, SE Linux (Chapter 9 of the Installation Manual) is initialized at this point. For a boot from 
the hard drive, the initialization occurs if an SE Linux filesystem has been saved to disk previously. 
For a PXE boot, the initialization takes place if the SELinuxInitialize directive is set to true in the node-installer.conf file.
5.4.11 Running Finalize Scripts

A finalize script is similar to an initialize script (section 5.4.5), only it runs a few stages later in the node-provisioning process.

In the context of configuration (section 3.15.4) it is used when custom commands need to be executed after the preceding mounting, provisioning, and housekeeping steps, but before handing over control to the node’s local init process. For example, custom commands may be needed to:

- initialize some not explicitly supported hardware before init takes over
- supply a configuration file for the software image that cannot simply be added to the software image and used by init because it needs node-specific settings
- load a slightly altered standard software image on particular nodes, typically with the change depending on automatically detecting the hardware of the node it is being loaded onto. While this could also be done by creating a full new software image and loading it on to the nodes according to the hardware, it usually turns out to be better for simplicity’s sake (future maintainability) to minimize the number of software images for the cluster.

The custom commands used to implement such changes are then added to the finalize script. How to edit a finalize script is described in Appendix E.2.

A finalize script can be added to both a node’s category and the node configuration. The node-installer first runs a finalize script, if it exists, from the node’s category, and then a finalize script, if it exists, from the node’s configuration.

The node-installer sets several environment variables which can be used by the finalize script. Appendix E contains an example script which documents these variables.

5.4.12 Unloading Specific Drivers

Many kernel drivers are only required during the installation of the node. After installation they are not needed and can degrade node performance.

Baseboard Management Controllers (BMCs, section 3.7) that use IPMI drivers are an egregious example of this. The IPMI drivers are required to have the node-installer configure the IP address of any IPMI cards. Once the node is configured, these drivers are no longer needed, but they continue to consume significant CPU cycles and power if they stay loaded, which can affect job performance.

To solve this, the node-installer can be configured to unload a specified set of drivers just before it hands over control to the local init process. This is done by editing the removeModulesBeforeInit setting in the node-installer configuration file /cm/node-installer/scripts/node-installer.conf. By default, the IPMI drivers are placed in the removeModulesBeforeInit setting.

To pick up IPMI-related data values, IPMI access is then carried out over the network without the drivers.

5.4.13 Switching To The Local init Process

At this point the node-installer is done. The node’s local drive now contains a complete Linux installation and is ready to be started. The node-installer hands over control to the local /sbin/init process, which continues the boot process and starts all runlevel services. From here on the boot process continues as if the machine was started from the drive just like any other regular Linux machine.

5.5 Node States

During the boot process, several state change messages are sent to the head node CMDaemon or detected by polling from the head node CMDaemon. The most important node states for a cluster after boot up are introduced in section 2.1.1. These states are described again, along with some less common ones to give a more complete picture of node states.

© Bright Computing, Inc.
5.5 Node States

5.5.1 Node States Icons In Bright View

In the node icons used by Bright View:

- Nodes in the **UP** state are indicated by an up-arrow.
  - If all health checks (section 12.2.4) for the node are successful, the up-arrow is green.
  - If there is a health check that fails or if the node requires a reboot, the up-arrow is red.

- Nodes in the **DOWN** state are indicated by a blue down-arrow.

- There are some other states, including:
  - Nodes in a **CLOSED** state are indicated by an X
  - Nodes in a **DOWN** state that are installing are indicated by a underscored down-arrow icon: ↓

5.5.2 Node States Shown In cmsh

In cmsh, the node state can be found using the `status` command from `device` mode for a node:

**Example**

```
[bright82->device]# status -n node001..node002
node001 ............. [ UP ] restart-required, health check failed
node002 ............. [ DOWN ] (hostname changed) restart-required
```

Devices in general can have their states conveniently listed with the `list -f` (page 28) command:

**Example**

```
[bright82->device]# list -f "hostname:10, status:48"
hostname ( status
---------- ------------------------------------------------
apc01 [ UP ]
bright82 [ UP ]
devhp [ UP ]
node001 [ UP ] restart-required, health check failed
node002 [ DOWN ] (hostname changed) restart-required
```

The reason for a red icon as shown in section 5.5.1 can be found within the parentheses. In this example it is (hostname changed).

5.5.3 Node States Indicating Regular Start Up

During a successful boot process the node goes through the following states:

- **INSTALLING.** This state is normally entered as soon as the node-installer has determined on which node the node-installer is running. Within this state, information messages display indicating what is being done while the node is in the INSTALLING state. Possible messages under the status column for the node within cmsh and Bright View are normally, in sequence:

  1. node-installer started
  2. Optionally, the following two messages:
     (a) waiting for user input
     (b) installation was resumed
  3. checking disks
  4. recreating partitions and filesystems
  5. mounting disks
6. One of these following two messages:
   (a) waiting for FULL provisioning to start
   (b) waiting for SYNC provisioning to start
7. provisioning started, waiting for completion
8. provisioning complete
9. initializing SELinux

Between steps 1 and 3 in the preceding, these optional messages can also show up:

- If burn mode is entered or left:
  running burn-in tests
  burn-in test completed successfully
- If maintenance mode is entered:
  entered maintenance mode

- INSTALLER_CALLINGINIT. This state is entered as soon as the node-installer has handed over control to the local init process. The associated message normally seen with it in cmsh and Bright View is:
  - switching to local root

- UP. This state is entered as soon as the CMDaemon of the node connects to the head node CMDaemon.

5.5.4 Node States That May Indicate Problems

Other node states are often associated with problems in the boot process:

- DOWN. This state is registered as soon as the CMDaemon on the regular node is no longer detected by CMDaemon on the head node. In this state, the state of the regular node is still tracked, so that CMDaemon is aware if the node state changes.

- CLOSED. This state is appended to the UP or DOWN state of the regular node by the administrator, and causes most CMDaemon monitoring actions for the node to cease. The state of the node is however still tracked by default, so that CMDaemon is aware if the node state changes.

The CLOSED state can be set from the device mode of cmsh using the close command. The help text for the command gives details on how it can be applied to categories, groups and so on. The -m option sets a message by the administrator for the closed node or nodes.

Example

```
root@headymcheadface ~]# cmsh
[headymcheadface]# device
[headymcheadface->device]# close -m "fan dead" -n node001,node009,node020
Mon May 2 16:32:01 [notice] headymcheadface: node001 ...[ DOWN/CLOSED ] (fan dead)
Mon May 2 16:32:01 [notice] headymcheadface: node009 ...[ DOWN/CLOSED ] (fan dead)
Mon May 2 16:32:01 [notice] headymcheadface: node020 ...[ DOWN/CLOSED ] (fan dead)
```

The CLOSED state can also be set from Bright View via the clickpath

Devices→Nodes→Edit↓Monitored state→Open/Close.

When the CLOSED state is set for a device, CMDaemon commands can still attempt to act upon it. For example, in the device mode of cmsh:
– open: This is the converse to the close command. It has the same options, including the -m option that logs a message. It also has the following extra options:
  * --reset: Resets whatever the status is of the devicestatus check. However, this reset by itself does not solve any underlying issue. The issue may still require a fix, despite the status having been reset. For example, the --reset option can be used to reset the restart-required flag (section 5.5.2). However, the reason that set the restart-required flag is not solved by the reset. Restarts are required for regular nodes if there have been changes in the following: network settings, disk setup, software image, or category.
  * -f|--failbeforedown <count>: Specifies the number of failed pings before a device is marked as down (default is 1).

– drain and undrain (Appendix G.4.1)
– For nodes that have power control:
  * power -f on
  * power -f off
  * power -f reset

In Bright View, the equivalents for a node node001 for example, are via the clickpaths:

- Devices→Nodes[node001]→Edit→Monitored state→Open/Close
- Devices→Nodes[node001]→Edit→Workload→Drain/Undrain
- Devices→Nodes[node001]→Edit→Power→On/Off/Reset.

CMDaemon on the head node only maintains device monitoring logs for a device that is in the UP state. If the device is in a state other than UP, then CMDaemon only tracks its state, and can display the state if queried.

For example: if a node displays the state UP when queried about its state, and is given a 'close' command, it then goes into a CLOSED state. Querying the node state then displays the state UP/CLOSED. It remains in that CLOSED state when the node is powered down. Querying the node state after being powered down displays DOWN/CLOSED. Next, powering up the node from that state, and having it go through the boot process, has the node displaying various CLOSED states during queries. Typically the query responses show it transitioning from DOWN/CLOSED, to INSTALLING/CLOSED, to INSTALLER_CALLINGINIT/CLOSED, and ending up displaying the UP/CLOSED state.

Thus, a node set to a CLOSED state remains in a CLOSED state regardless of whether the node is in an UP or DOWN state. The only way out of a CLOSED state is for the administrator to tell the node to open via the cmsh “open” option discussed earlier. The node, as far as CMDaemon is concerned, then switches from the CLOSED state to the OPEN state. Whether the node listens or not does not matter—the head node records it as being in an OPENING state for a short time, and during this time the next OPEN state (UP/OPEN, DOWN/OPEN, etc) is agreed upon by the head node and the node. When querying the state of a node, an OPEN tag is not displayed in the response, because it is the “standard” state. For example, UP is displayed rather than UP/OPEN. In contrast, a CLOSED tag is displayed when it is active, because it is a “special” state.

The CLOSED state is normally set to take a node that is unhealthy out of the cluster management system. The node can then still be in the UP state, displaying UP/CLOSED. It can even continue running workload jobs in this state, since workload managers run independent of CMDaemon. So, if the workload manager is still running, the jobs themselves are still handled by the workload manager, even if CMDaemon is no longer aware of the node state until the node is re-opened. For this reason, draining a node is often done before closing a node, although it is not obligatory.

\(^{3}\)power control mechanisms such as PDUs, custom power scripts, and BMCs using IPMI/HP iLO/DRAC/CIMC/Redfish, are described in Chapter 4

© Bright Computing, Inc.
• **OPENING.** This transitional state is entered as soon as the CMDaemon of the node rescinds the \texttt{CLOSED} state with an “open” command from \texttt{cmsh}. The state usually lasts no more than about 5 seconds, and never more than 30 seconds in the default configuration settings of Bright Cluster Manager. The help text for the \texttt{open} command of \texttt{cmsh} gives details on its options.

• **INSTALLER\_FAILED.** This state is entered from the \texttt{INSTALLING} state when the node-installer has detected an unrecoverable problem during the boot process. For instance, it cannot find the local drive, or a network interface cannot be started. This state can also be entered from the \texttt{INSTALLER\_CALLINGINIT} state when the node takes too long to enter the \texttt{UP} state. This could indicate that handing over control to the local \texttt{init} process failed, or the local \texttt{init} process was not able to start the CMDaemon on the node. Lastly, this state can be entered when the previous state was \texttt{INSTALLER\_REBOOTING} and the reboot takes too long.

• **INSTALLER\_UNREACHABLE.** This state is entered from the \texttt{INSTALLING} state when the head node CMDaemon can no longer ping the node. It could indicate the node has crashed while running the node-installer.

• **INSTALLER\_REBOOTING.** In some cases the node-installer has to reboot the node to load the correct kernel. Before rebooting it sets this state. If the subsequent reboot takes too long, the head node CMDaemon sets the state to \texttt{INSTALLER\_FAILED}.

## 5.6 Updating Running Nodes

Changes made to the contents of the software image for nodes, kept on the head node, become a part of any other provisioning nodes according to the housekeeping system on the head node (section 5.2.4).

Thus, when a regular node reboots, the latest image is installed from the provisioning system onto the regular node via a provisioning request (section 5.4.7).

However, updating a running node with the latest software image changes is also possible without rebooting it. Such an update can be requested using \texttt{cmsh} or Bright View, and is queued and delegated to a provisioning node, just like a regular provisioning request. The properties that apply to the regular provisioning an image also apply to such an update. For example, the value of the \texttt{provisioninginterface} setting (section 5.4.7) on the node being updated determines which interface is used to receive the image. In \texttt{cmsh} the request is submitted with the \texttt{imageupdate} option (section 5.6.2), while in Bright View, it is submitted, for a node \texttt{node001} for example, using the clickpath:

```
Devices \rightarrow Nodes[node001] \rightarrow Edit \rightarrow Software image \rightarrow Update node
```

The \texttt{imageupdate} command and “Update node” menu option use a configuration file called \texttt{excludelistupdate}, which is, as its name suggests, a list of exclusions to the update.

The \texttt{imageupdate} command and “Update node” menu option update what is on a running node from a stored image. The converse, that is, to update a stored image from what is on a running node, can be also be carried out. This converse can be viewed as grabbing from a node, and synchronizing what is grabbed, to an image. It can be done using \texttt{grabimage} (\texttt{cmsh}), or \texttt{Synchronize image} (Bright View), and involves further exclude lists \texttt{excludelistgrab} or \texttt{excludelistgrabnew}. The \texttt{grabimage} command and \texttt{Synchronize} option are covered in detail in section 11.5.2.

### 5.6.1 Updating Running Nodes: Configuration With \texttt{excludelistupdate}

The exclude list \texttt{excludelistupdate} used by the \texttt{imageupdate} command is defined as a property of the node’s category. It has the same structure and \texttt{rsync} patterns syntax as that used by the exclude lists for provisioning the nodes during installation (section 5.4.7).

Distinguishing Between The Intention Behind The Various Exclude Lists

The administrator should note that it is the \texttt{excludelistupdate} list that is being discussed here, in contrast with the \texttt{excludelistsyncinstall/excludelistfullinstall} lists which are discussed in section 5.4.7, and also in contrast with the \texttt{excludelistgrab/excludelistgrabnew} lists of section 11.5.2.
So, for the `imageupdate` command the `excludelistupdate` list concerns an *update* to a running system, while for installation sync or full provisioning, the corresponding exclude lists (`excludelistsyncinstall` and `excludelistfullinstall`) from section 5.4.7 are about an *install* during node start-up. Because the copying intention during updates is to be speedy, the `imageupdate` command synchronizes files rather than unnecessarily overwriting unchanged files. Thus, the `excludelistupdate` exclusion list it uses is actually analogous to the `excludelistsyncinstall` exclusion list used in the sync case of section 5.4.7, rather than being analogous to the `excludelistfullinstall` list.

Similarly, the `excludelistgrab`/`excludelistgrabnew` lists of section 11.5.2 are about a *grab* from the running node to the image. The `excludelistgrab` list is here intended for synchronizing the existing image with the running node, and is thus analogous to the `excludelistsyncinstall` exclusion list, while the `excludelistgrabnew` list is intended for copying a full new image from the running node, and is thus analogous to the `excludelistfullinstall` list.

The following table summarizes this:

<table>
<thead>
<tr>
<th>During:</th>
<th>Exclude list used is:</th>
<th>Copy intention:</th>
</tr>
</thead>
<tbody>
<tr>
<td>update</td>
<td><code>excludelistupdate</code></td>
<td>sync, image to running node</td>
</tr>
<tr>
<td>install</td>
<td><code>excludelistfullinstall</code></td>
<td>full, image to starting node</td>
</tr>
<tr>
<td></td>
<td><code>excludelistsyncinstall</code></td>
<td>sync, image to starting node</td>
</tr>
<tr>
<td>grab</td>
<td><code>excludelistgrabnew</code></td>
<td>full, running node to image</td>
</tr>
<tr>
<td></td>
<td><code>excludelistgrab</code></td>
<td>sync, running node to image</td>
</tr>
</tbody>
</table>

The preceding table is rather terse. It may help to understand it if it is expanded with some in-place footnotes, where the footnotes indicate what actions can cause the use of the exclude lists:

<table>
<thead>
<tr>
<th>During:</th>
<th>Exclude list used is:</th>
<th>Copy intention:</th>
</tr>
</thead>
<tbody>
<tr>
<td>update</td>
<td>eg: <code>imageupdate</code></td>
<td><code>excludelistupdate</code> sync, image to running node</td>
</tr>
<tr>
<td>install</td>
<td>eg: <code>node-provisioning</code> process during pre-init stage depending on installmode decision</td>
<td><code>excludelistfullinstall</code> full, image to starting node</td>
</tr>
<tr>
<td></td>
<td>eg: node provisioning with installmode FULL</td>
<td><code>excludelistsyncinstall</code> sync, image to starting node</td>
</tr>
<tr>
<td></td>
<td>eg: node provisioning AUTO with healthy partition</td>
<td><code>excludelistgrab</code> sync, running node to image</td>
</tr>
<tr>
<td>grab</td>
<td>eg: <code>grabimage (cmsh), grabimage -i/Grab to image (Bright View), and Grab to image (Bright View)</code></td>
<td><code>excludelistgrabnew</code> full, running node to image</td>
</tr>
<tr>
<td></td>
<td></td>
<td><code>excludelistgrab</code> sync, running node to image</td>
</tr>
</tbody>
</table>

© Bright Computing, Inc.
The Exclude List Logic For `excludelistupdate`

During an `imageupdate` command, the synchronization process uses the `excludelistupdate` list, which is a list of files and directories. One of the cross checking actions that may run during the synchronization is that the items on the list are excluded when copying parts of the filesystem from a known good software image to the node. The detailed behavior is as follows:

The `excludelistupdate` list is in the form of two sublists. Both sublists are lists of paths, except that the second sublist is prefixed with the text “no-new-files: ” (without the double quotes). For the node being updated, all of its files are looked at during an `imageupdate` synchronization run. During such a run, the logic that is followed is:

- if an excluded path from `excludelistupdate` exists on the node, then nothing from that path is copied over from the software image to the node
- if an excluded path from `excludelistupdate` does not exist on the node, then
  - if the path is on the first, non-prefixed list, then the path is copied over from the software image to the node.
  - if the path is on the second, prefixed list, then the path is not copied over from the software image to the node. That is, no new files are copied over, like the prefix text implies.

This is illustrated by figure 5.17.
The files and directories on the node that are not in the sub-lists lose their original contents, and are copied over from the software image. So, content not covered by the sub-lists at that time is normally not protected from deletion.

Thus, the provisioning system excludes paths described according to the `excludelistupdate` property.

The provisioning system also excludes a statically-imported filesystem on a node if the filesystem is a member of the following special list: NFS, Lustre, FUSE, CephFS, CIFS, PanFS, FhGFS, BeeGFS, GlusterFS, or GPFS. If this exclusion were not done, then all data on these imported filesystems would
be wiped, since they are not part of the software image. The automatic exclusion for these imported filesystems does not rely on the excludelist values maintained by CMDaemon—instead, CMDaemon carries out the check on-the-fly when provisioning starts.

Statically-imported filesystems that have their mounts managed by Bright Cluster Manager via the fsmounts mode can be excluded from being mounted on the nodes in the first place, by removing them from the listed mounts within the fsmounts mode.

Imported filesystems not on the special list can have their data wiped out during provisioning or sync updates, if the statically-imported filesystems are placed in the image manually—that is, if the filesystems are mounted manually into the image on the head node via /etc/fstab without using cmsh or Bright View.

Filesystems mounted dynamically cannot have their appearance or disappearance detected reliably: Any filesystem that may be imported via an auto-mount operation must therefore explicitly be excluded by the administrator manually adding the filesystem to the exclude list. This is to prevent an incorrect execution of imageupdate. Neglecting to do this may wipe out the filesystem, if it happens to be mounted in the middle of an imageupdate operation.

**The fstab system is a statically mounting system, and not an auto-mounter:** While fstab mounts filesystems automatically, system administrators should not confuse that with auto-mounting. Auto-mounting as provided by autofs is designed for the dynamic mounting of filesystems on demand by regular users. The fstab table is designed for mounting as carried out by the system, as occurs during boot, which is why it is regarded as a static, non-auto-mounting system.

### Editing An Exclude List

A sample cmsh one-liner which opens up a text editor in a category so that the exclude list for updates can be edited is:

```cmsh
cmsh -c "category use default; set excludelistupdate; commit"
```

Similarly, the exclude list for updates can also be edited in Bright View via the clickpath:

1. Go to Grouping → Node categories → Edit → Settings → Exclude list update

### Provisioning Modifications Via excludelistmanipulatescript

Sometimes the administrator has a need to slightly modify the execution of exclude lists during provisioning. The excludelistmanipulatescript file takes as an input the exclude list inherited from a category, modifies it in some way, and then produces a new exclude list. Conceptually it is a bit like how an administrator might use sed if it worked without a pipe. As usual, setting it for node level overrides the category level.

A script that manipulates the exclude lists of a node can be specified as follows within cmsh:

```bash
[bright82]% device use node001
[bright82->device[node001]]% set excludelistmanipulatescript
(a vi session will start. A script is edited and saved)
[bright82->device[node001*]]% commit
```

The script can be as simple as:

**Example**

```bash
#!/bin/bash

echo "- *"
echo 'no-new-files: - *'
```

© Bright Computing, Inc.
If provisioning a node from the head node, then the script modifies the node-provisioning exclude lists—`excludelistfullinstall`, `excludelistsyncinstall`, and `excludelistupdate`—so that they appear to contain these items only:

- *
  no-new-files: - *

The provisioning node then excludes everything during provisioning.

Careful editing and testing of the script is advised. Saving a script with just a single whitespace, for example, usually has undesirable results.

A more useful script template is the following:

**Example**

```bash
#!/bin/bash

while read; do
  echo "$REPLY"
done

echo "# This and next line added by category excludelistmanipulatescript."
echo "# The command line arguments were: $@
```

The provisioning exclude lists are simply read in, then sent out again without any change, except that the last two lines add comment lines to the exclude lists that are to be used.

Internally, the arguments taken by the `excludelistmanipulatescript` are the destination path and the sync mode (one of `install`|`update`|`full`|`grab`|`grabnew`). This can be seen in the output of `$@`, if running an `imageupdate` command to execute a dry run with the preceding example:

```
[bright82]% device use node001
[bright82->device[node001]]% get excludelistmanipulatescript
   (the script is put in)
[bright82->device[node001*]]% commit; imageupdate
Performing dry run (use synclog command to review result, then pass -w to perform real update)... Wed Apr 15 04:55:46 2015 [notice] bright82: Provisioning started: sending bright82:/cm/images/default-image to node001:/, mode UPDATE, dry run = yes, no data changes!
[bright82->device[node001]]% 
[bright82->device[node001*]]% commit; imageupdate
   (the script is put in)
[bright82->device[node001*]]% commit; imageupdate
Performing dry run (use synclog command to review result, then pass -w to perform real update)... Wed Apr 15 04:55:51 2015 [notice] bright82: Provisioning completed: sending bright82:/cm/images/default-image to node001:/, mode UPDATE, dry run = yes, no data changes!
imageupdate [ COMPLETED ]
```

An excerpt from the sync log, after running the `synclog` command, then shows output similar to (some output elided):

```
...
- /cm/shared/*
- /cm/shared/
- /home/*
- /home/
- /cm/shared/apps/slurm/*
- /cm/shared/apps/slurm/
# This and next line added by category excludelistmanipulatescript.
# The command line arguments were: update /
```
Rsync output:
sending incremental file list
cm/local/apps/cmd/scripts/healthchecks/configfiles/
...

Here, the sync mode is *update* and the destination path is "/". Which of the exclude lists is being modified can be determined by the `excludelistmanipulatescript` by parsing the sync mode.

The bash variable that accepts the exclude list text is set to a safely-marked form using curly braces. This is done to avoid expansion surprises, due to wild card characters in the exclude lists. For example, if `$REPLY` were used instead of `${REPLY}`, and the script were to accept an exclude list line containing "/proc/*", then it would give quite confusing output.

**Other Exclude List Handling Options**

The `excludelistfailover` and `excludelistnormal` files: are two further exclude list files that modify standard provisioning behavior. These are discussed in section 14.4.8.

The `cm-manipulate-internal-exclude-lists` tool: When synchronizing to a cloud director, or to an edge director, it is sometimes useful to exclude unneeded files and paths from the synchronization, in order to speed it up. The `excludelistmanipulatescript` tool is powerful enough to do it, but it has some issues due to its power. For example, it is a script, which means that is called whenever it is used, and so uses up some extra resources. Also, it is a bit tricky to set up.

An easier way to manipulate exclude lists for the unneeded files and paths is via the `cm-manipulate-internal-exclude-lists` tool, described in section 4.4.2 of the *Cloudbursting Manual*. This tool hardcodes a list using simple options.

**Exclude List State At Node Level**

Exclude lists at category level and node level: An exclude list can be set at node level, as well as at category level. Roles and overlays can add implied exclude lists too.

At category level, an exclude list such as `excludelistfullinstall` can be set up explicitly with:

**Example**

```
[bright82->category[default]]% set excludelistfullinstall
... a text editor such as vi opens up and the list can be edited...
[bright82->category[default]]% commit
```

At node level, an exclude list can be set in the same way:

**Example**

```
[bright82->device[node001]]% set excludelistfullinstall
... a text editor such as vi opens up and the list can be edited...
[bright82->device[node001]]% commit
```

An exclude list that is not empty at node level overrules its corresponding category list. Exclude lists brought in via roles are however simply included in the exclude list.

**The `excludelist` command:** At node level it can be unclear what the resulting exclude list (“operational exclude list”) actually is. The exclude list state at node level can therefore be viewed using the `excludelist` command options. The `excludelist` command becomes active if a software image has been set at the node level.

- The `list` option to `excludelist` lists the source and destination paths:
5.6 Updating Running Nodes

Example

[bright82->device[node001]]% excludelist list
Source path (on the head node) Destination path (on the node)
-------------------------------- --------------------------------
/cm/images/default-image /

- The get option to the excludelist command has synchronization mode and destination suboptions for a node.

Earlier on (page 194), the intention behind the various exclude lists, according to the type of update or synchronization, were distinguished.

The excludelist get command can have a destination path specified, and have the type of update or synchronization specified according to those distinguishing concepts.

The output to the excludelist get command then shows the operational exclude list as seen by a node for that path and for that update or synchronization.

Thus, for example:

- The full install operational exclude list for the path / on node node001, intended for a full installation to a node that is starting up, can be found as follows:

Example

[bright82->device[node001]]% excludelist get full /
# For details on the exclude patterns defined here please refer to
# the FILTER RULES section of the rsync man page.
#
# Files that match these patterns will not be installed onto the node.
# - lost+found/
# - /proc/*
# - /sys/*
# - /boot/efi
#
# extra defaults
# - /proc/*
# - /sys/*

- Similarly, the sync install operational exclude list for the path / on node node001, intended for a sync installation to a node that is starting up, can be found as follows:

Example

[bright82->device[node001]]% excludelist get sync /
# For details on the exclude patterns defined here please refer to
# the FILTER RULES section of the rsync man page.
#
# Files that exist on a node and match one of these patterns will not be
# modified or deleted. Any files that match one of these patterns and that
# exist in the image but are absent on the node, will be copied to the node.
# - /.autofsck
# - /boot/grub*/grub.cfg
# - /cm/local/apps/openldap/etc/certs/ldap.key
# - /cm/local/apps/openldap/etc/certs/ldap.pem
# - /data/*
# - /home/*

© Bright Computing, Inc.
Other excludeList get options, besides full and sync, are:

* grab (a grab from a running node for a sync back to an existing image)
* grabnew (a grab from a running node for a full install to a new image)
* update (a sync update of a running node from an image).

All excludeList get options correspond to the intentions of the associated exclude list types as distinguished on page 194.

5.6.2 Updating Running Nodes: With cmsh Using imageupdate

Using a defined excludeListUpdate property (section 5.6.1), the imageupdate command of cmsh is used to start an update on a running node:

Example

[bright82->device]% imageupdate -n node001
Performing dry run (use synclog command to review result, then pass -w to perform real update)...
Tue Jan 11 12:13:33 2011 bright82: Provisioning started on node node001
[bright82->device]% imageupdate -n node001: image update in progress ...
[bright82->device]%
Tue Jan 11 12:13:44 2011 bright82: Provisioning completed on node node001

By default the imageupdate command performs a dry run, which means no data on the node is actually written. Before passing the "-w" switch, it is recommended to analyze the rsync output using the synclog command (section 5.4.7).

If the user is now satisfied with the changes that are to be made, the imageupdate command is invoked again with the "-w" switch to implement them:

Example

[bright82->device]% imageupdate -n node001 -w
Provisioning started on node node001
node001: image update in progress ...
[bright82->device]% Provisioning completed on node node001

5.6.3 Updating Running Nodes: With Bright View Using the Update node Option

In Bright View, an image update can be carried out by selecting the specific node or category, for example node001, and updating it via the clickpath:
Devices→Nodes[n001]→Edit↓Software image→Update node

5.6.4 Updating Running Nodes: Considerations

An attempt to update the image on a running node can run into some issues:

- Updating an image via cmsh or Bright View automatically updates the provisioners first via the updateprovisioners command (section 5.2.4) if the provisioners have not been updated in the last 5 minutes. The conditional update period can be set with the provisioningnodeautoupdatetimeout parameter (section 5.2.4).

So, with the default setting of 5 minutes, if there has been a new image created within the last 5 minutes, then provisioners do not get the updated image when doing the updates, which means that nodes in turn do not get those updates. Running the updateprovisioners command just before running the imageupdate command therefore usually makes sense.

- By default, Bright Cluster Manager does not allow provisioning if automount (page 657) is running.
• Also, when updating services, the services on the nodes may not restart since the init process may not notice the replacement.

For these reasons, especially for more extensive changes, it can be safer for the administrator to simply reboot the nodes instead of using `imageupdate` to provision the images to the nodes. A reboot by default ensures that a node places the latest image with an AUTO install (section 5.4.7), and restarts all services.

The Reinstall node option, which can be run, for example, on a node node001, using a clickpath of Devices → Nodes[node001] → Edit → Software image → Reinstall node also does the same as a reboot with default settings, except for that it unconditionally places the latest image with a FULL install, and so may take longer to complete.

5.7 Adding New Nodes

How the administrator can add a single node to a cluster is described in section 1.3 of the Installation Manual. This section explains how nodes can be added in ways that are more convenient for larger numbers of nodes.

5.7.1 Adding New Nodes With cmsh And Bright View Add Functions

Node objects can be added from within the device mode of cmsh by running the `add` command:

Example

```
[bright82->device]% add physicalnode node002 10.141.0.2
[bright82->device][node002]% commit
```

The Bright View equivalent of this is following the clickpath:

Devices → Nodes → Add → Settings → Hostname

then adding the value node002 to Hostname, and saving it.

When adding the node objects in cmsh and Bright View, some values (IP addresses for example) may need to be filled in before the object validates. For regular nodes, there should be an interface and an IP address for the network that it boots from, as well as for the network that manages the nodes. A regular node typically has only one interface, which means that the same interface provides boot and management services. This interface is then the boot interface, BOOTIF, during the pre-init stage, but is also the management interface, typically eth0 or whatever the device is called, after the pre-init stage. The IP address for BOOTIF is normally provided via DHCP, while the IP address for the management interface is set to a static IP address that is set via cmsh or Bright View by the administrator.

Adding new node objects as “placeholders” can also be done from cmsh or Bright View. By placeholders, here it is meant that an incomplete node object is set. For example, sometimes it is useful to create a node object with the MAC address setting unfilled because it is still unknown. Why this can be useful is covered shortly.

5.7.2 Adding New Nodes With The Node Creation Wizard

Besides adding nodes using the `add` command of cmsh or the Add button of Bright View as in the preceding text, there is also a Bright View wizard that guides the administrator through the process—the node creation wizard. This is useful when adding many nodes at a time. It is available via the clickpath:

Devices → Nodes → Create nodes

This wizard should not be confused with the closely-related node identification resource described in section 5.4.2, which identifies unassigned MAC addresses and switch ports, and helps assign them node names.

• The node creation wizard creates an object for nodes, assigns them node names, but it leaves the MAC address field for these nodes unfilled, keeping the node object as a “placeholder”.

• The node identification resource assigns MAC addresses so that node names are associated with a MAC address.
If a node is left with an unassigned MAC address—that is, in a “placeholder” state—then it means that when the node starts up, the provisioning system lets the administrator associate a MAC address and switch port number at the node console for the node. This occurs when the node-installer reaches the node configuration stage during node boot as described in section 5.4.2. This is sometimes preferable to associating the node name with a MAC address remotely with the node identification resource.

The node creation wizard sets IP addresses for the nodes. By default it starts IP addressing for the new nodes by guessing an appropriate value from the node names. The administrator can override this starting value in the main screen of the wizard by modifying it in the header section of the IP addresses column (figure 5.18).

![Node Creation Wizard: Setting Interfaces](image)

Figure 5.18: Node Creation Wizard: Setting Interfaces

The cmsh equivalent of the node creation wizard is the foreach loop with the –clone option acting on a node (section 2.5.5).

5.8 Troubleshooting The Node Boot Process

During the node boot process there are several common issues that can lead to an unsuccessful boot. This section describes these issues and their solutions. It also provides general hints on how to analyze boot problems.

Before looking at the various stages in detail, the administrator may find that simply updating software or firmware may fix the issue. In general, it is recommended that all available updates are deployed on a cluster.

- Updating software is covered in Chapter 11.
  
  - On the head node, the most relevant software can be updated with `yum`, `zypper`, or `apt`, as explained in section 11.2. For example, with `yum`:
    
    **Example**
    ```
    yum update cmdaemon node-installer
    ```
  
  - Similarly for the software image, the most relevant software can be updated too. This is done via a procedure involving a chroot installation., as described in section 11.4. If using `yum`, then the update can be carried out within the image, `<software image>`, with:
    
    **Example**
    ```
    yum update --installroot=/cm/images/<software image> cmdaemon node-installer-slave
    ```
5.8 Troubleshooting The Node Boot Process

- UEFI or BIOS firmware should be updated as per the vendor recommendation

The various stages that may fail during node boot are now examined.

5.8.1 Node Fails To PXE Boot

Possible reasons to consider if a node is not even starting to network boot (PXE boot for x86 nodes) in the first place:

- DHCP may not be running. A check can be done to confirm that DHCP is running on the internal network interface (usually eth0):

```
[root@bright82 ~]# ps u -C dhcpd
USER PID %CPU %MEM VSZ RSS TTY STAT START TIME COMMAND
root 2448 0.0 0.0 11208 436 ? Ss Jan22 0:05 /usr/sbin/dhcpd eth0
```

This may indicate that Node booting is disabled in Bright View (figure 3.5, page 62) and needs to be enabled. The equivalent in cmsh is to check if the response to:

```
cmsh -c "network use internalnet; get nodebooting"
```

needs to be set to yes.

- The DHCP daemon may be “locked down” (section 3.2.1, figure 3.5, table 3.2.1). New nodes are granted leases only after lockdowntdhcpd is set to no in cmsh, or Lock down dhcpd is disabled in Bright View for the network.

- A rogue DHCP server may be running. If there are all sorts of other machines on the network the nodes are on, then it is possible that there is a rogue DHCP server active on it, perhaps on an IP address that the administrator has forgotten, and interfering with the expected PXE booting. Such stray DHCP servers should be eliminated.

  In such a case, removing all the connections and switches and just connecting the head node directly to a problem node, NIC-to-NIC, should allow a normal network boot to happen. If a normal network boot then does happen, it indicates the problem is indeed due to a rogue DHCP server on the more-connected network.

- The boot sequence may be set wrongly in the BIOS. The boot interface should normally be set to be the first boot item in the BIOS.

- The node may be set to boot from UEFI mode. If UEFI mode has a buggy network boot implementation, then it may fail to network boot. For x86 nodes, setting the node to PXE boot using the legacy BIOS mode can be tried instead, or perhaps the UEFI firmware can be updated.

- There may a bad cable connection. This can be due to moving the machine, or heat creep, or another physical connection problem. Firmly inserting the cable into its slot may help. Replacing the cable or interface as appropriate may be required.

- There may a problem with the switch. Removing the switch and connecting a head node and a regular node directly with a cable can help troubleshoot this.

  Disabling the Spanning Tree Protocol (STP) functions of a managed switch is recommended. With STP on, nodes may randomly fail to network boot.

- The cable may be connected to the wrong interface. By default, eth0 is normally assigned the internal network interface, and eth1 the external network interface on the head node for a type 1 network. However:
The two interfaces can be confused when physically viewing them and a connection to the wrong interface can therefore be made.

It is also possible that the administrator has changed the default assignment.

If the node is booting from iPXE (that is from a hard drive), for the very first time, and there is more than one interface on the node, then the interface assignment is not guaranteed in Red Hat and derivatives before version 7.

The node running a version before RHEL7 can therefore end up unable to pick up the provisioning image from the interface that is in use for provisioning. Moving the provisioning cable to the correct interface on the node at this very first boot resolves this issue.

### Interface Naming Conventions Post-RHEL7

https://access.redhat.com/documentation/en-US/Red_Hat_Enterprise_Linux/7/html/Networking_Guide/ch-Consistent_Network_Device_Naming.html describes the consistent network device scheme for interfaces in RHEL7. This scheme sets an interface assignment on iPXE boot for multiple interfaces that is also valid by default during the very first iPXE boot. This means that an administrator can know which interface is used for provisioning and can connect the provisioning cable accordingly.

### Reverting To The Pre-RHEL7 Interface Naming Conventions

To revert to the pre-RHEL7 behavior, the text:

```
net.ifnames=0 biosdevname=0
```

can be appended to the line starting with `GRUB_CMDLINE_LINUX` in `/etc/default/grub` within the head node. For this:

* The `biosdevname` parameter only works if the dev helper is installed. The dev helper is available from the `biosdevname` RPM package. The parameter also requires that the system supports SMBIOS 2.6 or ACPI DSM.
* The `net.ifnames` parameter is needed if `biosdevname` is not installed.

**Example**

```
GRUB_CMDLINE_LINUX="rd.lvm.lv=centos/swap vconsole.keymap=us \
crashkernel=auto rd.lvm.lv=centos/root vconsole.font=latarcyr\ 
heb-sun16 rhgb quiet net.ifnames=0 biosdevname=0"
```

A cautious system administrator may back up the original `grub.cfg` file:

```
[root@bright82 ~]# cp --preserve /boot/grub2/grub.cfg /boot/grub2/grub.cfg.orig
```

The GRUB configuration should be generated with:

```
[root@bright82 ~]# grub2-mkconfig -o /boot/grub2/grub.cfg
```

For a regular node, the text `net.ifnames=0 biosdevname=0` can be appended to the `kernelparameters` property, for an image selected from `softwareimage` mode.

**Example**

```
[bright82->softwareimage]% list
Name (key)             Path
----------------------- ------------------------------
default-image         /cm/images/default-image
openstack-image       /cm/images/openstack-image
[bright82->softwareimage]% use default-image
[bright82->softwareimage[default-image]]% append kernelparameters " net.ifnames=0 biosdevname=0"
[bright82->softwareimage*[default-image*]]% commit
```
The append command requires a space at the start of the quote, in order to separate the kernel parameters from any pre-existing ones.

The connections should be checked to eliminate these possibilities.

- The TFTP server that sends out the image may have hung. During a normal run, an output similar to this appears when an image is in the process of being served:

```
[root@bright82 ~]# ps ax | grep [t]ftp
    7512 ?  Ss  0:03 in.tftpd --maxthread 500 /tftpboot
```

If the TFTP server is in a zombie state, the head node should be rebooted. If the TFTP service hangs regularly, there is likely a networking hardware issue that requires resolution.

Incidentally, grepping the process list for a TFTP service returns nothing when the head node is listening for TFTP requests, but not actively serving a TFTP image. This is because the TFTP service runs under xinet.d and is called on demand. Running

```
[root@bright82 ~]# chkconfig --list
```

should include in its output the line:

```
tftp: on
```

if TFTP is running under xinet.d.

- The xinetd server on the head node that manages the TFTP service in RHEL6 may be hung. If so, then it should be restarted.

- The switchover process from TFTP to HTTP may have hung. During a normal provisioning run, assuming that CMDaemon uses the default AdvancedConfig setting of PXEBootOverHTTP=1 (Appendix C), TFTP is used to load the initial boot loader, but the kernel and ramdisk are loaded up via HTTP for speed. Some hardware has problems with switching over to using HTTP.

In that case, setting PXEBootOverHTTP=0 keeps the node using TFTP for loading the kernel and ramdisk, and should work. Another possible way to solve this is to upgrade the PXE boot BIOS to a version that does not have this problem.

- Sometimes a manufacturer releases hardware with buggy drivers that have a variety of problems. For instance: Ethernet frames may be detected at the interface (for example, by `ethtool`), but TCP/IP packets may not be detected (for example, by `wireshark`). In that case, the manufacturer should be contacted to upgrade their driver.

- The interface may have a hardware failure. In that case, the interface should be replaced.

### 5.8.2 Node-installer Logging

If the node manages to get beyond the net booting stage to the node-installer stage, then the first place to look for hints on node boot failure is usually the node-installer log file. The node-installer runs on the node that is being provisioned, and sends logging output to the syslog daemon running on that node. This forwards all log data to the IP address from which the node received its DHCP lease, which is typically the IP address of the head node or failover node. In a default Bright Cluster Manager setup, the `local5` facility of the syslog daemon is used on the node that is being provisioned to forward all node-installer messages to the log file `/var/log/node-installer` on the head node.

After the node-installer has finished running, its log is also stored in `/var/log/node-installer` on the regular nodes.

If there is no node-installer log file anywhere yet, then it is possible that the node-installer is not yet deployed on the node. Sometimes this is due to a system administrator having forgotten to change a
provisioning-related configuration setting. One possibility is that the nodegroups setting (section 5.2.1), if used, may be misconfigured. Another possibility is that the image was set to a locked state (section 5.4.7). The provisioningstatus -a command can indicate this:

Example

[bright82->softwareimage]# provisioningstatus -a | grep locked
Scheduler info: requested software image is locked, request deferred

To get the image to install properly, the locked state should be removed for a locked image.

Example

[root@bright82 ~]# cmsh -c "softwareimage foreach * (get name; get locked)"
default-image yes
[root@bright82 ~]# cmsh -c "softwareimage; foreach * (set locked no); commit"
[root@bright82 ~]# cmsh -c "softwareimage foreach * (get name; get locked)"
default-image no

The node automatically picks up the image after it is unlocked.

Optionally, extra log information can be written by enabling debug logging, which sets the syslog importance level at LOG_DEBUG. To enable debug logging, the debug field is changed in /cm/node-installer/scripts/node-installer.conf.

From the console of the booting node the log file is generally accessible by pressing Alt+F7 on the keyboard. Debug logging is however excluded from being viewed in this way, due to the output volume making this impractical.

A booting node console can be accessed remotely if Serial Over LAN (SOL) is enabled (section 13.7), to allow the viewing of console messages directly. A further depth in logging can be achieved by setting the kernel option loglevel=N, where N is a number from 0 (KERN_EMERG) to 7 (KERN_DEBUG).

One possible point at which the node-installer can fail on some hardware is if SOL (section 13.7) is enabled in the BIOS, but the hardware is unable to cope with the flow. The installation can freeze completely at that point. This should not be confused with the viewing quirk described in section 13.7.4, even though the freeze typically appears to take place at the same point, that point being when the console shows "freeing unused kernel memory" as the last text. One workaround to the freeze would be to disable SOL.

5.8.3 Provisioning Logging

The provisioning system sends log information to the CMDaemon log file. By default this is in /var/log/cmdaemon on the local host, that is, the provisioning host. The host this log runs on can be configured with the CMDaemon directive SyslogHost (Appendix C).

The image synchronization log file can be retrieved with the synclog command running from device mode in cmsh. Hints on provisioning problems are often found by looking at the tail end of the log.

If the tail end of the log shows an rsync exit code of 23, then it suggests a transfer error. Sometimes the cause of the error can be determined by examining the file or filesystem for which the error occurs. For the rsync transport, logs for node installation are kept under /var/spool/cmd/, with a log written for each node during provisioning. The name of the node is set as the prefix to the log name. For example node002 generates the log:

/var/spool/cmd/node002-.rsync

© Bright Computing, Inc.
5.8 Troubleshooting The Node Boot Process

5.8.4 Ramdisk Fails During Loading Or Sometime Later

One issue that may come up after a software image update via yum, zypper, or apt (section 11.4), is that the ramdisk stage may fail during loading or sometime later, for a node that is rebooted after the update. This occurs if there are instructions to modify the ramdisk by the update. In a normal machine the ramdisk would be regenerated. In a cluster, the extended ramdisk that is used requires an update, but Bright Cluster Manager is not aware of this. Running the `createramdisk` command from cmsh or the `Recreate Initrd` command via the clickpaths:

- Devices → Nodes → Edit → Kernel → Recreate Initrd
- Grouping → Node Categories → Edit → Kernel → Recreate Initrd
- Provisioning → Software Images → Edit → Recreate Initrd

(section 5.3.2) generates an updated ramdisk for the cluster, and solves the failure for this case.

Another, somewhat related possible cause of a halt at this stage, is that the kernel modules that are to be loaded may have been specified at wrongly by the administrator in the hierarchy of software image, category, or node (page 159). A check of the kernel modules specified in softwareimage mode, category mode, or device mode (for the particular node) may reveal a misconfiguration.

5.8.5 Ramdisk Cannot Start Network

The ramdisk must activate the node’s network interface in order to fetch the node-installer. To activate the network device, the correct kernel module needs to be loaded. If this does not happen, booting fails, and the console of the node displays something similar to figure 5.19.

![Figure 5.19: No Network Interface](image)

To solve this issue the correct kernel module should be added to the software image’s kernel module configuration (section 5.3.2). For example, to add the e1000 module to the default image using cmsh:

**Example**
Node Provisioning

Node-Installer Cannot Create Disk Layout

When the node-installer is not able to create a drive layout it displays a message similar to figure 5.20. The node-installer log file (section 5.8.2) contains something like:

```
Mar 24 13:55:31 10.141.0.1 node-installer: Installmode is: AUTO
Mar 24 13:55:31 10.141.0.1 node-installer: Checking partitions and
filesystems.
Mar 24 13:55:32 10.141.0.1 node-installer: Detecting device '/dev/sda':
not found
Mar 24 13:55:32 10.141.0.1 node-installer: Detecting device '/dev/hda':
not found
Mar 24 13:55:32 10.141.0.1 node-installer: Partitions and/or filesystems
are missing/corrupt. (Exit code 4, signal 0)
Mar 24 13:55:32 10.141.0.1 node-installer: Detecting device '/dev/sda':
not found
Mar 24 13:55:32 10.141.0.1 node-installer: Detecting device '/dev/hda':
not found
(Exit code 4, signal 0)
Mar 24 13:55:32 10.141.0.1 node-installer: There was a fatal problem. This node can not be
installed until the problem is corrected.
```
Disk layout failures can have several reasons.

**BIOS And Order Issues**
One reason may be that the drive may be disabled in the BIOS. It should be enabled.

Another reason may be that the drive order was changed. This could happen if, for example, a defective motherboard has been replaced. The drive order should be kept the same as it was before a motherboard change.

**Read-only Mode Issues**
Another reason may be due to SSDs that have a hardware jumper or toggle switch that sets a drive to read-only mode. A read-only mode drive will typically fail at this point. The drive should be made writeable.

**Hardware Issues**
If the node-installer log for the node shows lines with the text *Input/output error*, then it generally indicates a hardware issue. Possible hardware issues include:

- a drive failure
- a faulty cable between storage and controller
- a faulty storage controller
- a faulty backplane in the server

If the node has enough RAM, then it is possible to boot up the node up as a diskless node, to carry out further diagnosis with disk tools such as `smartmontools`.

**Software Driver Issues**
One of the most common software issues is that the correct storage driver is not being loaded. To solve this issue, the correct kernel module should be added to the software image’s kernel module configuration (section 5.3.2).
Experienced system administrators work out what drivers may be missing by checking the results of hardware probes. For example, going into the node-installer shell using Alt-F2, and then looking at the output of lspci, shows a list of hardware detected in the PCI slots and gives the chipset name of the storage controller hardware in this case:

Example

```
[installer> root@node001 ~]# lspci | grep SCSI
00:10.0 Serial Attached SCSI controller: LSI Logic / Symbios Logic SAS2
  008 PCI-Express Fusion-MPT SAS-2 [Falcon] (rev 03)
```

The next step is to Google with likely search strings based on that output.

The Linux Kernel Driver DataBase (LKDDb) is a hardware database built from kernel sources that lists driver availability for Linux. It is available at http://cateee.net/lkddb/. Using the Google search engine’s "site" operator to restrict results to the cateee.net web site only, a likely string to try might be:

Example

SAS2008 site:cateee.net

The search result indicates that the mpt2sas kernel module needs to be added to the node kernels. A look in the modules directory of the software image shows if it is available:

Example

```
find /cm/images/default-image/lib/modules/ -name "*mpt2sas*"
```

If it is not available, the driver module must then be obtained. If it is a source file, it will need to be compiled. By default, nodes run on standard distribution kernels, so that only standard procedures need to be followed to compile modules.

If the module is available, it can be added to the default image, by using cmsh in softwareimage mode to create the associated object. The object is given the same name as the module, i.e. mp2sas in this case:

Example

```
[bright82] % softwareimage use default-image
[bright82->softwareimage[default-image]]% kernelmodules
[bright82->softwareimage[default-image]->kernelmodules]% add mpt2sas
[bright82->softwareimage[default-image]->kernelmodules[mpt2sas]]% commit
Thu May 19 16:54:52 2011 [notice] bright82: Initial ramdisk for image default-image was being generated
Thu May 19 16:55:43 2011 [notice] bright82: Initial ramdisk for image default-image was regenerated successfully.
```

After committing the change it can take some time before ramdisk creation is completed—typically about a minute, as the example shows. Once the ramdisk is created, the module can be seen in the list displayed from kernelmodules mode. On rebooting the node, it should now continue past the disk layout stage.

© Bright Computing, Inc.
5.8.7 Node-Installer Cannot Start BMC (IPMI/iLO) Interface

In some cases the node-installer is not able to configure a node’s BMC interface, and displays an error message similar to figure 5.21.

![No BMC Interface](image)

**Figure 5.21: No BMC Interface**

Usually the issue can be solved by adding the correct BMC (IPMI/iLO) kernel modules to the software image’s kernel module configuration. However, in some cases the node-installer is still not able to configure the BMC interface. If this is the case the BMC probably does not support one of the commands the node-installer uses to set specific settings, or there may be a hardware glitch in the BMC.

**The setupBmc Node-Installer Configuration Setting**

To solve this issue, setting up BMC interfaces can be disabled globally by setting the `setupBmc` field in the node-installer configuration file `/cm/node-installer/scripts/node-installer.conf` to false. Doing this disable configuration of all BMC interfaces by the node-installer. A custom finalize script (Appendix E) can then be used to run the required commands instead.

The `setupBmc` field in the node-installer should not be confused with the `SetupBMC` directive in `cmd.conf` (Appendix C). The former is about enabling the BMC interface, while the latter is about enabling automated passwords to the BMC interface (an interface that must of course be enabled in the first place to work).

**The failOnMissingBmc Node-Installer Configuration Setting**

If the kernel modules for the BMC are loaded up correctly, and the BMC is configured, but it is not detected by the node-installer, then the node-installer halts by default. This corresponds to the setting `failOnMissingBmc = true` in the node-installer configuration file `/cm/node-installer/scripts/node-installer.conf`. Toggling this to `false` skips BMC network device detection, and lets the node-installer continue past the BMC detection and configuration stage. This can be convenient, for example, if the BMC is not yet configured and the aim is to get on with setting up the rest of the cluster.

© Bright Computing, Inc.
The failOnFailedBmcCommand Node-Installer Configuration Setting
If a BMC command fails, then the node-installer by default terminates node installation. The idea behind this is to allow the administrator to fix the problem. Sometimes, however, hardware can wrongly signal a failure. That is, it can signal a false failure, as opposed to a true failure.

A common case is the case of ipmitool. ipmitool is used by Bright Cluster Manager to configure the BMC. With most hardware vendors it works as expected, signaling success and failure correctly. As per the default behavior: with success, node installation proceeds, while with failure, it terminates.

With certain hardware vendors however ipmitool fails with an exit code 1, even though the BMC is properly configured. Again, as per the default behavior: success has node installation proceed, while failure has node installation terminate. Only this time, because the failure signal is incorrect, the termination on failure is also incorrect behavior.

To get around the default behavior for false failure cases, the administrator can force the node-installer to set the value of failOnFailedBmcCommand to false in the node-installer configuration file /cm/node-installer/scripts/node-installer.conf. The installation then skips past the false failure.

BMC Hardware Glitch And Cold Reset
Sometimes, typically due to a hardware glitch, a BMC can get into a state where it is not providing services, but the BMC is still up (responding to pings). Contrariwise, a BMC may not respond to pings, but still respond to IPMI commands. A fix for such glitchy states is usually to power cycle the BMC. This is typically done, either physically, or by using a BMC management tool such as ipmitool.

Physically resetting the power supply to the BMC is done typically by pulling the power cable out and then pushing it in again. For typical rack-based servers the server can just be pulled out and in again. Just doing a shutdown of the server with the power cable still in place normally does not power down the BMC.

BMC management does allow the BMC to power down and be reset from software, without having to physically handle the server. This software-based cold reset is a BIOS-manufacturer-dependent feature. A popular tool used for managing BMCs that can do such a cold reset is ipmitool. This can be run remotely, but also on the node console if the node cannot be reached remotely.

With ipmitool, a cold reset is typically carried out with a command such as:

```bash
[root@bright82 ~]# module load ipmitool
[root@bright82 ~]# ipmitool -U <bmcusername> -P <bmcpassword> -H <host IP> -I lanplus mc reset cold
```

The values for <bmcusername> and <bmcpassword> can be obtained as shown in section 3.7.2.

BMC Troubleshooting With The System Event Log
The System Event Log (SEL) can be read with:

```bash
[root@bright82 ~]# module load ipmitool
[root@bright82 ~]# ipmitool -U <bmcusername> -P <bmcpassword> -H <host IP> -I lanplus sel list
```

The timestamped output can be inspected for errors related to the CPU, ECC, or memory.

Other BMC Troubleshooting
Some more specific commands for handling IPMI might be via the service ipmi <option> commands, which can show the IPMI service has failed to start up:

Example

```bash
[root@bright82 ~]# service ipmi status
Redirecting to /bin/systemctl status ipmi.service
ipmi.service - IPMI Driver
Loaded: loaded (/usr/lib/systemd/system/ipmi.service; disabled; vendor preset: enabled)
Active: inactive (dead)
```

© Bright Computing, Inc.
In the preceding session the driver has simply not been started up. It can be started up with the start option:

Example

[root@bright82 ~]# service ipmi start
Redirecting to /bin/systemctl start ipmi.service
Job for ipmi.service failed because the control process exited with error code. See "systemctl status ipmi.service" and "journalctl -xe" for details.

In the preceding session, the start up failed. The service status output shows:

Example

[root@bright82 ~]# service ipmi status -l
Redirecting to /bin/systemctl status -l ipmi.service
ipmi.service - IPMI Driver
Loaded: loaded (/usr/lib/systemd/system/ipmi.service; disabled; vendor preset: enabled)
Active: failed (Result: exit-code) since Mon 2016-12-19 14:34:27 CET; 2min 3s ago
Process: 8930 ExecStart=/usr/libexec/openipmi-helper start (code=exited, status=1/FAILURE)
Main PID: 8930 (code=exited, status=1/FAILURE)

Dec 19 14:34:27 bright82 systemd[1]: Starting IPMI Driver...
Dec 19 14:34:27 bright82 openipmi-helper[8930]: Startup failed.
Dec 19 14:34:27 bright82 systemd[1]: ipmi.service: main process exited, code=exited, status=1/FAILURE
Dec 19 14:34:27 bright82 systemd[1]: Failed to start IPMI Driver.
Dec 19 14:34:27 bright82 systemd[1]: Unit ipmi.service entered failed state.
Dec 19 14:34:27 bright82 systemd[1]: ipmi.service failed.

Further details can be found in the journal:

Example

[root@bright82 ~]# journalctl -xe | grep -i ipmi
...
-- Unit ipmi.service has begun starting up.
Dec 19 14:34:27 bright82 kernel: ipmi message handler version 39.2
Dec 19 14:34:27 bright82 kernel: IPMI System Interface driver.
Dec 19 14:34:27 bright82 kernel: ipmi_si: Unable to find any System Interface(s)
Dec 19 14:34:27 bright82 openipmi-helper[8930]: Startup failed.
...

In the preceding session, the failure is due to a missing BMC interface (Unable to find any System Interface(s). A configured BMC interface should show an output status similar to:

Example

[root@bright82 ~]# service ipmi status
Redirecting to /bin/systemctl status ipmi.service
ipmi.service - IPMI Driver
Loaded: loaded (/usr/lib/systemd/system/ipmi.service; disabled; vendor preset: enabled)
Active: active (exited) since Mon 2016-12-19 14:37:10 CET; 2min 0s ago
Process: 61019 ExecStart=/usr/libexec/openipmi-helper start (code=exited, status=0/SUCCESS)
Main PID: 61019 (code=exited, status=0/SUCCESS)
Dec 19 14:37:10 bright82 systemd[1]: Starting IPMI Driver...
Dec 19 14:37:10 bright82 systemd[1]: Started IPMI Driver.
Sometimes the issue may be an incorrect networking specification for the BMC interfaces. MAC and IP details that have been set for the BMC interface can be viewed with the `lan print` option to `ipmitool` if the service has been started:

**Example**

```
[root@bright82 ~]# module load ipmitool
[root@bright82 ~]# ipmitool lan print
Set in Progress : Set Complete
Auth Type Support : MD5 PASSWORD
Auth Type Enable : Callback : MD5 PASSWORD
               : User : MD5 PASSWORD
               : Operator : MD5 PASSWORD
               : Admin : MD5 PASSWORD
               : OEM :
IP Address Source : Static Address
IP Address : 93.184.216.34
Subnet Mask : 255.255.255.0
MAC Address : aa:bb:01:02:cd:ef
SNMP Community String : public
IP Header : TTL=0x00 Flags=0x00 Precedence=0x00 TOS=0x00
BMC ARP Control : ARP Responses Enabled, Gratuitous ARP Disabled
Gratituous ARP Intrvl : 0.0 seconds
Default Gateway IP : 93.184.216.1
Default Gateway MAC : 00:00:00:00:00:00
Backup Gateway IP : 0.0.0.0
Backup Gateway MAC : 00:00:00:00:00:00
802.1q VLAN ID : Disabled
802.1q VLAN Priority : 0
RMCP+ Cipher Suites : 0,1,2,3,4,6,7,8,9,11,12,13,15,16,17,18
Cipher Suite Priv Max : caaaaaaaaaaaaaa
               : X=Cipher Suite Unused
               : c=CALLBACK
               : u=USER
               : o=OPERATOR
               : a=ADMIN
               : 0=OEM

During normal operation the metrics (Appendix G) displayed by Bright Cluster Manager are useful. However, if those are not available for some reason, then the direct output from BMC sensor metrics may be helpful for troubleshooting:

**Example**

```
[root@bright82 ~]# module load ipmitool
[root@bright82 ~]# ipmitool sensor list all
# ipmitool sensor list
Ambient Temp | 22.000 | degrees C | ok | na | na | na | 38.000 | 41.000 | 45.000
AVG Power | 300.000 | Watts | ok | na | na | na | na | na |
Fan 1 Tach | 4125.000 | RPM | ok | na | na | na | na | na |
...
User Management

Users and groups for the cluster are presented to the administrator in a single system paradigm. That is, if the administrator manages them with the Bright Cluster Manager, then the changes are automatically shared across the cluster (the single system).

Bright Cluster Manager runs its own LDAP service to manage users, rather than using unix user and group files. In other words, users and groups are managed via the centralizing LDAP database server running on the head node, and not via entries in /etc/passwd or /etc/group files.

Sections 6.1 and 6.2 cover the most basic aspects of how to add, remove and edit users and groups using Bright Cluster Manager.

Section 6.3 describes how an external LDAP server can be used for authentication services instead of the one provided by Bright Cluster Manager.

Section 6.4 discusses how users can be assigned only selected capabilities when using Bright View or cmsh, using profiles with sets of tokens.

6.1 Managing Users And Groups With Bright View

Within Bright View:

- users can be managed via the clickpath Identity Management → Users
- groups can be managed via the clickpath Identity Management → Groups.

For users (figure 6.1) the LDAP entries for regular users are displayed. These entries are editable and each user can then be managed in further detail.

There is already one user on a newly installed Bright Cluster Manager: cmsupport. This user has no password set by default, which means (section 6.2.2) no logins to this account are allowed by default. Bright Cluster Manager uses the user cmsupport to run various diagnostics utilities, so it should not be removed, and the default contents of its home directory should not be removed.

There are five buttons, Revert, Add, Delete, Save, and Edit available in the window:

1. Add: allows users to be added via a dialog (figure 6.2). These additions can be committed via the Save button.
An explanation of the less obvious items in the dialog follows:

- **Shadow warning**: The number of days, before the password expires, that the user is warned of the expiry
- **Shadow max**: The maximum number of days the password is valid
- **Shadow min**: The minimum number of days required between password changes. A value of
zero means the user may change their password at any time

- **Inactive**: The number of days of inactivity allowed for the user before the account is blocked. A value of zero means the user is never blocked

- **Profile**: The preconfigured capability that the user is assigned. Available settings are:
  - `admin`: Allows the user to run Bright View with the same privileges as user `admin`
  - `cloudjob`: Allows the user to run `cmsub`, the cloud job submission utility (section 4.7 of the *User Manual*, and section 4.3 of the *Cloudbursting Manual*)
  - `portal`: Allows the user to access the user portal
  - `readonly`: Allows the user to run Bright View without the ability to modify settings.
  - `none`: (default). Prevents the user from using Bright View

A profile setting only takes effect if the certificate for the user is used. User certificates are only persistent for a cluster with a permanent license (page 76 of the *Installation Manual*), so the administrator should check the license is not a temporary license before attempting to use this feature. Section 6.4 explains the concepts of capabilities, profiles, certificates, and tokens.

2. **Save**: saves the as-yet-uncommitted `Add` or `Edit` operations. When saving an addition:
   - User and group ID numbers are automatically assigned from UID and GID 1000 onwards, except for RHEL6 and derivatives which assign from 500 onwards.
   - A home directory is created and a login shell is set. Users with unset passwords cannot log in.

3. **Edit**: allows user attributes to be modified via a dialog similar to the `Add` dialog of figure 6.2.

4. **Revert**: discards unsaved edits that have been made via the `Edit` button. The reversion goes back to the last save.

5. **Delete**: deletes selected rows of users. By default, their home directories are not deleted.

Group management in Bright View is carried out via the clickpath `Identity Management → Groups`. Clickable LDAP object entries for regular groups then show up, similar to the user entries already covered. Management of these entries is done with the same button functions as for user management.

### 6.2 Managing Users And Groups With cmsh

User management tasks as carried out by Bright View in section 6.1, can be carried with the same end results in cmsh too.

A cmsh session is run here in order to cover the functions corresponding to the user management functions of Bright View of section 6.1. These functions are run from within the `user` mode of cmsh:

**Example**

```
[root@bright82 ]# cmsh
[bright82]% user
[bright82->user]%
```

#### 6.2.1 Adding A User

This part of the session corresponds to the functionality of the `Add` button operation in section 6.1. In user mode, the process of adding a user `maureen` to the LDAP directory is started with the `add` command:

**Example**

```
[bright82->user]# add maureen
[bright82->user*[maureen*]]%
```
The cmsh utility helpfully drops into the user object just added, and the prompt shows the user name to reflect this. Going into user object would otherwise be done manually by typing use maureen at the user mode level.

Asterisks in the prompt are a helpful reminder of a modified state, with each asterisk indicating that there is an unsaved, modified property at that asterisk’s level.

The modified command displays a list of modified objects that have not yet been committed:

**Example**

```
[bright82->user*[maureen*]]% modified
State Type Name
  + User maureen
```

This corresponds roughly to what is displayed by the Unsaved entities icon in the top right corner of the Bright View standard display (figure 12.5).

Running show at this point reveals a user name entry, but empty fields for the other properties of user maureen. So the account in preparation, while it is modified, is clearly not yet ready for use:

**Example**

```
[bright82->user*[maureen*]]% show
Parameter Value
Common name  maureen
Expiration date 2038/1/1
```

### 6.2.2 Saving The Modified State

This part of the session corresponds to the functionality of the Save button operation in section 6.1.

In section 6.2.1 above, user maureen was added. maureen now exists as a proposed modification, but has not yet been committed to the LDAP database.

Running the commit command now at the maureen prompt stores the modified state at the user maureen object level:

**Example**

```
[bright82->user*[maureen*]]% commit
[bright82->user[maureen]]% show
Parameter Value
Common name maureen
Expiration date 2038/1/1
```
Group ID 1002
Home directory /home/maureen
Inactive 0
Last change 2011/5/30
Login shell /bin/bash
Password *********
Profile
Revision
Shadow max 999999
Shadow min 0
Shadow warning 7
User ID 1002
User name maureen

If, however, commit were to be run at the user mode level without dropping into the maureen object level, then instead of just that modified user, all modified users would be committed.

When the commit is done, all the empty fields for the user are automatically filled in with defaults based the underlying Linux distribution used. Also, as a security precaution, if an empty field (that is, a “not set”) password entry is committed, then a login into the account is not allowed. So, in the example, the account for user maureen exists at this stage, but still cannot be logged into until the password is set. Editing passwords and other properties is covered in section 6.2.3.

The default permissions for file and directories under the home directory of the user are defined by the umask settings in /etc/login.defs, as would be expected if the administrator were to use the standard useradd command. Setting a path for the homedirectory parameter for a user sets a default home directory path. By default the default path is /home/<username> for a user <username>. If homedirectory is unset, then the default is determined by the HomeRoot directive (Appendix C).

6.2.3 Editing Properties Of Users And Groups
This corresponds roughly to the functionality of the Edit button operation in section 6.1.

In the preceding section 6.2.2, a user account maureen was made, with an unset password as one of its properties. Logins to accounts with an unset password are refused. The password therefore needs to be set if the account is to function.

Editing Users With set And clear
The tool used to set user and group properties is the set command. Typing set and then either using tab to see the possible completions, or following it up with the enter key, suggests several parameters that can be set, one of which is password:

Example
[bright82->user[maureen]]% set

Name:
set - Set specific user property

Usage:
set [user] <parameter> <value> [...][value] ... (type 1)
set [user] <parameter> [file] (type 2)

Arguments:
user
userID of the user, omit if current is set

Parameters: (type 1)
Revision ............ Entity revision

© Bright Computing, Inc.
commonname .......... Full user name
e-mail .............. email
expirationdate ...... Indicates the date on which the user login will be disabled
groupid ............. Base group of this user
homedirectory ...... Home directory
inactive ............. Indicates the number of days of inactivity allowed for the user
kubernetesrolebindings
loginshell .......... Login shell
password ............ Password
profile ............. Profile for Authorization
shadowmax ........... Indicates the maximum number of days for which the user password remains valid.
shadowmin ........... Indicates the minimum number of days required between password changes
shadowwarning ...... The number of days of advance warning given to the user before the user password expires
surname ............. Surname
userid .............. User id number
username ............ User name

Parameters: (type 2)
notes ............... Administrator notes

Continuing the session from the end of section 6.2.2, the password can be set at the user context prompt like this:

**Example**

[bright82->user[maureen]]% set password seteca5tr0n0my
[bright82->user*[maureen*]]% commit
[bright82->user[maureen]]%

At this point, the account maureen is finally ready for use.

The converse of the set command is the clear command, which clears properties:

**Example**

[bright82->user[maureen]]% clear password; commit

**Editing Groups With** append And removefrom

While the above commands set and clear also work with groups, there are two other commands available which suit the special nature of groups. These supplementary commands are append and removefrom. They are used to add extra users to, and remove extra users from a group.

For example, it may be useful to have a printer group so that several users can share access to a printer. For the sake of this example (continuing the session from where it was left off in the preceding), tim and fred are now added to the LDAP directory, along with a group printer:

**Example**

[bright82->user[maureen]]% add tim; add fred
[bright82->user*[fred*]]% exit; group; add printer
[bright82->group*[printer*]]% commit
[bright82->group[printer]]% exit; exit; user
[bright82->user*]%
The context switch that takes place in the preceding session should be noted: The context of user maureen was eventually replaced by the context of group printer. As a result, the group printer is committed, but the users tim and fred are not yet committed, which is indicated by the asterisk at the user mode level.

Continuing onwards, to add users to a group the append command is used. A list of users maureen, tim and fred can be added to the group printer like this:

Example

```
[bright82->user*]% commit
Successfully committed 2 Users
[bright82->user%] group use printer
[bright82->group[printer]]% append groupmembers maureen tim fred; commit
[bright82->group[printer]]% show
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Group ID</td>
<td>1003</td>
</tr>
<tr>
<td>Group members</td>
<td>maureen tim fred</td>
</tr>
<tr>
<td>Group name</td>
<td>printer</td>
</tr>
</tbody>
</table>

To remove users from a group, the removefrom command is used. A list of specific users, for example, tim and fred, can be removed from a group like this:

```
[bright82->group[printer]]% removefrom groupmembers tim fred; commit
[bright82->group[printer]]% show
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Group ID</td>
<td>1003</td>
</tr>
<tr>
<td>Group members</td>
<td>maureen</td>
</tr>
<tr>
<td>Group name</td>
<td>printer</td>
</tr>
</tbody>
</table>

The clear command can also be used to clear members—but it also clears all of the extras from the group:

Example

```
[bright82->group[printer]]% clear groupmembers
[bright82->group*[printer*]]% show
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Group ID</td>
<td>1003</td>
</tr>
<tr>
<td>Group members</td>
<td></td>
</tr>
<tr>
<td>Group name</td>
<td>printer</td>
</tr>
</tbody>
</table>

The commit command is intentionally left out at this point in the session in order to illustrate how reversion is used in the next section.

### 6.2.4 Reverting To The Unmodified State

This corresponds roughly to the functionality of the Revert button operation in section 6.1.

This section (6.2.4) continues on from the state of the session at the end of section 6.2.3. There, the state of group printers was cleared so that the extra added members were removed. This state (the state with no group members showing) was however not yet committed.

The refresh command reverts an uncommitted object back to the last committed state.

This happens at the level of the object it is using. For example, the object that is being handled here is the properties of the group object printer. Running revert at a higher level prompt—say, in the group mode level—would revert everything at that level and below. So, in order to affect only the properties
of the group object printer, the refresh command is used at the group object printer level prompt. It then reverts the properties of group object printer back to their last committed state, and does not affect other objects:

Example

```
[bright82->group*printer*]% refresh
[bright82->group[printer]]% show
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Group ID</td>
<td>1003</td>
</tr>
<tr>
<td>Group members</td>
<td>maureen</td>
</tr>
<tr>
<td>Group name</td>
<td>printer</td>
</tr>
</tbody>
</table>

Here, the user maureen reappears because she was stored in the last save. Also, because only the group object printer has been committed, the asterisk indicates the existence of other uncommitted, modified objects.

6.2.5 Removing A User

Removing a user using cmsh corresponds roughly to the functionality of the Delete button operation in section 6.1. The remove command removes a user or group. The useful “-d|--data” flag added to the end of the username removes the user’s home directory too. For example, within user mode, the command “remove user maureen -d; commit” removes user maureen, along with her home directory. Continuing the session at the end of section 6.2.4 from where it was left off, as follows, shows this result:

Example

```
[bright82->group[printer]]% user use maureen
[bright82->user[maureen]]% remove -d; commit
Successfully removed 1 Users
Successfully committed 0 Users
[bright82->user]% !ls -d /home/*| grep maureen #no maureen left behind
[bright82->user]%
```

6.3 Using An External LDAP Server

Sometimes, an external LDAP server is used to serve the user database. If, instead of just using the database for authentication, the user database is also to be managed, then its LDAP schema must match the Bright Cluster Manager LDAP schema.

For RHEL7, the `/etc/nslcd.conf`, `/etc/openldap/ldap.conf`, and the certificate files under `/cm/local/apps/openldap/etc/certs/` should be copied over. Port 636 on ShoreWall running on the head node should be open for LDAP communication over the external network, if external nodes are using it on the external network. In addition, the external nodes and the head node must be able to resolve each other.

By default, Bright Cluster Manager runs an LDAP health check using the cmsupport user on the LDAP server. The LDAP health check may need to be modified or disabled by the administrator to prevent spurious health warnings with an external LDAP server:

**Modifying Or Disabling The ldap Healthcheck**

**Modifying the ldap health check:** To keep a functional ldap health check with an external LDAP server, a permanent external LDAP user name, for example ldapcheck, can be added. This user can then be set as the parameter for Bright Cluster Manager’s ldap health check object that is used to monitor the LDAP service. Health checks and health check objects are discussed in Chapter 12.
6.3 Using An External LDAP Server

- If user management is not configured to work on CMDaemon for the external LDAP server, then the user management tool that is used with the external LDAP server should be used by the administrator to create the ldapcheck user instead.

- If user management is still being done via CMDaemon, then an example session for configuring the ldap script object to work with the new external LDAP user is (some prompt text elided):

**Example**

```bash
[root@bright82 ~]# cmsh
[bright82]# user
[bright82->user]# add ldapcheck; commit
[bright82->user[ldapcheck]]# monitoring setup ldap
[bright82->monitoring->setup[ldap]]# show
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Arguments</td>
<td></td>
</tr>
</tbody>
</table>

```
[bright82->monitoring->setup[ldap]]# set arguments "ldapcheck"; commit
[bright82->monitoring->setup[ldap]]#
```

**Disabling the ldap health check:** Instead of modifying the ldap health check to work when using an external LDAP server, it can be disabled entirely via Bright View or cmsh.

- Bright View: the ldap health check is disabled via the clickpath:

  Monitoring→Data Producers→ldap→Edit

- cmsh: the disabled parameter of the ldap health check object is set to yes. The disabled parameter for the ldap health check can be set as follows:

  ```bash
  [root@bright82 ~]# cmsh -c "monitoring setup use ldap; set disabled yes; commit"
  ```

**Configuring The Cluster To Authenticate Against An External LDAP Server**

The cluster can be configured in different ways to authenticate against an external LDAP server. For smaller clusters, a configuration where LDAP clients on all nodes point directly to the external server is recommended. An easy way to set this up is as follows:

- On the head node:
  - In distributions that are:
    - derived from prior to RHEL 6: the URLs in /etc/ldap.conf, and in the image file /cm/images/default-image/etc/ldap.conf are set to point to the external LDAP server.
    - derived from the RHEL 6.x series: the file /etc/ldap.conf does not exist. The files in which the changes then need to be made are /etc/nslcd.conf and /etc/pam_ldap.conf. To implement the changes, the nslcd daemon must then be restarted, for example with service nslcd restart.
    - derived from RHEL 7.x series: the file /etc/ldap.conf does not exist. The files in which the changes then need to be made are /etc/nslcd.conf and /etc/openldap/ldap.conf. To implement the changes, the nslcd daemon must then be restarted, for example with service nslcd restart.
    - the updateprovisioners command (section 5.2.4) is run to update any other provisioners.
Then, the configuration files are updated in the software images that the nodes use. If the nodes use the default-image, and if the nodes are based on RHEL7 and derivatives, then the files to update are /cm/images/default-image/etc/nslcd.conf and /cm/images/default/etc/openldap/ldap.conf. After the configuration change has been made, and the nodes have picked up the new configuration, the regular nodes can then carry out LDAP lookups.

- Nodes can simply be rebooted to pick up the updated configuration, along with the new software image.
- Alternatively, to avoid a reboot, the imageupdate command (section 5.6.2) can be run to pick up the new software image from a provisioner.

- The CMDaemon configuration file cmd.conf (Appendix C) has LDAP user management directives. These may need to be adjusted:
  - If another LDAP tool is to be used for external LDAP user management instead of Bright View or cmsh, then altering cmd.conf is not required, and Bright Cluster Manager’s user management capabilities do nothing in any case.
  - If, however, system users and groups are to be managed via Bright View or cmsh, then CMDaemon, too, must refer to the external LDAP server instead of the default LDAP server. This configuration change is actually rare, because the external LDAP database schema is usually an existing schema generated outside of Bright Cluster Manager, and so it is very unlikely to match the Bright Cluster Manager LDAP database schema. To implement the changes:
    * On the node that is to manage the database, which is normally the head node, the LDAPHost, LDAPUser, LDAPPass, and LDAPSearchDN directives in cmd.conf are changed so that they refer to the external LDAP server.
    * CMDaemon is restarted to enable the new configurations.

For larger clusters the preceding solution can cause issues due to traffic, latency, security and connectivity fault tolerance. If such occur, a better solution is to replicate the external LDAP server onto the head node, hence keeping all cluster authentication local, and making the presence of the external LDAP server unnecessary except for updates. This optimization is described in the next section.

### 6.3.1 External LDAP Server Replication

This section explains how to set up replication for an external LDAP server to an LDAP server that is local to the cluster, if improved LDAP services are needed. Section 6.3.2 then explains how this can then be made to work with a high availability setup.

Typically, the Bright Cluster Manager LDAP server is configured as a replica (consumer) to the external LDAP server (provider), with the consumer refreshing its local database at set timed intervals. How the configuration is done varies according to the LDAP server used. The description in this section assumes the provider and consumer both use OpenLDAP.

**External LDAP Server Replication: Configuring The Provider**

It is advisable to back up any configuration files before editing them.

The provider is assumed to be an external LDAP server, and not necessarily part of the Bright Cluster Manager cluster. The LDAP TCP ports 389 and 6389 may therefore need to be made accessible between the consumer and the provider by changing firewall settings.

If a provider LDAP server is already configured then the following synchronization directives must be in the slapd.conf file to allow replication:

```bash
index entryCSN eq
index entryUUID eq
overlay syncprov
syncprov-checkpoint <ops> <minutes>
syncprov-sessionlog <size>
```

© Bright Computing, Inc.
The openldap documentation (http://www.openldap.org/doc/) has more on the meanings of these directives. If the values for <ops>, <minutes>, and <size> are not already set, typical values are:

```
syncprov-checkpoint 1000 60
```
and:

```
syncprov-sessionlog 100
```

To allow the consumer to read the provider database, the consumer’s access rights need to be configured. In particular, the userPassword attribute must be accessible. LDAP servers are often configured to prevent unauthorized users reading the userPassword attribute.

Read access to all attributes is available to users with replication privileges. So one way to allow the consumer to read the provider database is to bind it to replication requests.

Sometimes a user for replication requests already exists on the provider, or the root account is used for consumer access. If not, a user for replication access must be configured.

A replication user, syncuser with password secret can be added to the provider LDAP with adequate rights using the following syncuser.ldif file:

```
dn: cn=syncuser,<suffix>
objectClass: person
cn: syncuser
sn: syncuser
userPassword: secret
```

Here, <suffix> is the suffix set in slapd.conf, which is originally something like dc=example,dc=com. The syncuser is added using:

```
ldapadd -x -D "cn=root,<suffix>" -W -f syncuser.ldif
```

This prompts for the root password configured in slapd.conf.

To verify syncuser is in the LDAP database the output of ldapsearch can be checked:

```
ldapsearch -x "(sn=syncuser)"
```

To allow access to the userPassword attribute for syncuser the following lines in slapd.conf are changed, from:

```
access to attrs=userPassword
  by self write
  by anonymous auth
  by * none
```
to:

```
access to attrs=userPassword
  by self write
  by dn="cn=syncuser,<suffix>" read
  by anonymous auth
  by * none
```

Provider configuration is now complete and the server can be restarted using service ldap restart, or for RHEL 7.x: service slapd restart.

© Bright Computing, Inc.
External LDAP Server Replication: Configuring The Consumer(s)

The consumer is an LDAP server on a Bright Cluster Manager head node. It is configured to replicate with the provider by adding the following lines to `/cm/local/apps/openldap/etc/slapd.conf`:

```
syncrepl rid=2
  provider=ldap://external.ldap.server
  type=refreshOnly
  interval=01:00:00:00
  searchbase=<suffix>
  scope=sub
  schemachecking=off
  binddn="cn=syncuser,<suffix>"
  bindmethod=simple
  credentials=secret
```

Here:

- The `rid=2` value is chosen to avoid conflict with the `rid=1` setting used during high availability configuration (section 6.3.2).
- The `provider` argument points to the external LDAP server.
- The `interval` argument (format DD:HH:MM:SS) specifies the time interval before the consumer refreshes the database from the external LDAP. Here, the database is updated once a day.
- The `credentials` argument specifies the password chosen for the `syncuser` on the external LDAP server.

More on the `syncrepl` directive can be found in the openldap documentation (http://www.openldap.org/doc/).

The configuration files must also be edited so that:

- The `<suffix>` and `rootdn` settings in `slapd.conf` both use the correct `<suffix>` value, as used by the provider.
- The `base` value in `/etc/ldap.conf` uses the correct `<suffix>` value as used by the provider. This is set on all Bright Cluster Manager nodes including the head node(s). If the `ldap.conf` file does not exist, then the note on page 225 about RHEL versions applies.

Finally, before replication takes place, the consumer database is cleared. This can be done by removing all files, except for the `DB_CONFIG` file, from under the configured database directory, which by default is at `/var/lib/ldap/`.

The consumer is restarted using `service ldap restart`. This replicates the provider’s LDAP database, and continues to do so at the specified intervals.

6.3.2 High Availability

No External LDAP Server Case

If the LDAP server is not external—that is, if the Bright Cluster Manager is set to its high availability configuration, with its LDAP servers running internally, on its own head nodes—then by default LDAP services are provided from both the active and the passive node. The high-availability setting ensures that CMDaemon takes care of any changes needed in the `slapd.conf` file when a head node changes state from passive to active or vice versa, and also ensures that the active head node propagates its LDAP database changes to the passive node via a `syncprov/syncrepl` configuration in `slapd.conf`. 

© Bright Computing, Inc.
External LDAP Server With No Replication Locally Case
In the case of an external LDAP server being used, but with no local replication involved, no special high-availability configuration is required. The LDAP client configuration in /etc/ldap.conf simply remains the same for both active and passive head nodes, pointing to the external LDAP server. The file /cm/images/default-image/etc/ldap.conf, in each software image also point to the same external LDAP server. If the ldap.conf files referred to here in the head and software images do not exist, then the note on page 225 about RHEL versions applies.

External LDAP Server With Replication Locally Case
In the case of an external LDAP server being used, with the external LDAP provider being replicated to the high-availability cluster, it is generally more efficient for the passive node to have its LDAP database propagated and updated only from the active node to the passive node, and not updated from the external LDAP server.

The configuration should therefore be:

- an active head node that updates its consumer LDAP database from the external provider LDAP server
- a passive head node that updates its LDAP database from the active head node’s LDAP database

Although the final configuration is the same, the sequence in which LDAP replication configuration and high availability configuration are done has implications on what configuration files need to be adjusted.

1. For LDAP replication configuration done after high availability configuration, adjusting the new suffix in /cm/local/apps/openldap/etc/slapd.conf and in /etc/ldap.conf on the passive node to the local cluster suffix suffices as a configuration. If the ldap.conf file does not exist, then the note on page 225 about RHEL versions applies.

2. For high availability configuration done after LDAP replication configuration, the initial LDAP configurations and database are propagated to the passive node. To set replication to the passive node from the active node, and not to the passive node from an external server, the provider option in the syncrepl directive on the passive node must be changed to point to the active node, and the suffix in /cm/local/apps/openldap/etc/slapd.conf on the passive node must be set identical to the head node.

The high availability replication event occurs once only for configuration and database files in Bright Cluster Manager’s high availability system. Configuration changes made on the passive node after the event are therefore persistent.

6.4 Tokens And Profiles

Tokens can be assigned by the administrator to users so that users can carry out some of the operations that the administrator do with Bright View or cmsh. Every cluster management operation requires that each user, including the administrator, has the relevant tokens in their profile for the operation.

The tokens for a user are grouped into a profile, and such a profile is typically given a name by the administrator according to the assigned capabilities. For example the profile might be called readmonitoringonly if it allows the user to read the monitoring data only, or it may be called powerhandler if the user is only allowed to carry out power operations. Each profile thus consists of a set of tokens, typically relevant to the name of the profile, and is typically assigned to several users.

The profile is stored as part of the authentication certificate (section 2.3) which is generated for running authentication operations to the cluster manager for the certificate owner.

Profiles are handled with the profiles mode of cmsh, or from the Profiles window, accessible via a clickpath of Identity Management→Profiles
The following preconfigured profiles are available from cmsh:

<table>
<thead>
<tr>
<th>Profile name</th>
<th>Default Tasks Allowed</th>
</tr>
</thead>
<tbody>
<tr>
<td>admin</td>
<td>all tasks</td>
</tr>
<tr>
<td>cloudjob</td>
<td>cloud job submission</td>
</tr>
<tr>
<td>cmhealth</td>
<td>health-related prejob tasks</td>
</tr>
<tr>
<td>cmpam</td>
<td>Bright Cluster Manager PAM tasks</td>
</tr>
<tr>
<td>node</td>
<td>node-related</td>
</tr>
<tr>
<td>portal</td>
<td>user portal viewing</td>
</tr>
<tr>
<td>power</td>
<td>device power</td>
</tr>
<tr>
<td>readonly</td>
<td>view-only</td>
</tr>
</tbody>
</table>

The available preconfigured profiles in cmsh can be seen by running the list command in profile mode. Another way to see them is by using tab-completion prompting, as follows:

Example

```
[root@bright82 ~]# cmsh
[bright82%] profile
[bright82->profile]% show <TAB> <TAB>
admin cloudjob cmhealth cmpam node portal power readonly
```

The tokens, and other properties of a particular profile can be seen within profile mode as follows:

Example

```
[bright82->profile]% show readonly
Parameter     Value
-------------- ---------------------------------...
Name          readonly
Non user       no
Revision       
Services       CMDevice CMNet CMPart CMMon CMJob CMAuth CMServ CMUser CMSession CMMain CMGui CMP+
Tokens         GET_DEVICE_TOKEN GET_CATEGORY_TOKEN GET_NODEGROUP_TOKEN POWER_STATUS_TOKEN GET_DE+
```

For screens that are not wide enough to view the parameter values, the values can also be listed:

Example

```
[bright82->profile]% get readonly tokens
GET_DEVICE_TOKEN
GET_CATEGORY_TOKEN
GET_NODEGROUP_TOKEN
```

A profile can be set with cmsh for a user within user mode as follows:

Example

```
[root@bright82 ~]# cmsh
[bright82%] user use conner
[bright82->user[conner]]% get profile
[bright82->user[conner]]% set profile readonly; commit
```
6.4.1 Modifying Profiles

A profile can be modified by adding or removing appropriate tokens to it. For example, the readonly group by default has access to the burn status and burn log results. Removing the appropriate tokens stops users in that group from seeing these results.

In cmsh the removal can be done from within profile mode as follows:

```
[root@bright82 ~]# cmsh
[bright82]# profile use readonly
[...[readonly]]# removefrom tokens burn_status_token get_burn_log_token
[bright82]# ->profile*[readonly*]# commit
```

Tab-completion after typing in `removefrom tokens` helps in filling in the tokens that can be removed.

In Bright View (figure 6.3), the same removal action can be carried out via the clickpath:

Identity Management → Profiles → readonly → Edit → Tokens

In the resulting display it is convenient to maximize the window. Also convenient is running a search for burn, which will show the relevant tokens, BURN_STATUS_TOKEN and GET_BURN_LOG_TOKEN, as well as the subgroup they are in device. The ticks can be removed from the BURN_STATUS_TOKEN and GET_BURN_LOG_TOKEN checkboxes, and the changed settings can then be saved.

![Figure 6.3: Bright View Profile Token Management](image)

6.4.2 Creation Of Custom Certificates With Profiles, For Users Managed By Bright Cluster Manager's Internal LDAP

Custom profiles can be created to include a custom collection of capabilities in cmsh and Bright View. Cloning of profiles is also possible from cmsh.

A certificate file, with an associated expiry date, can be created based on a profile. The time of expiry for a certificate cannot be extended after creation. An entirely new certificate is required after expiry of the old one.

The creation of custom certificates using cmsh (page 233) or Bright View (page 234) is described later on. After creating such a certificate, the openssl utility can be used to examine its structure and properties. In the following example most of the output has been elided in order to highlight the expiry date (30 days from the time of generation), the common name (democert), the key size (2048), profile properties (readonly), and system login name (peter), for such a certificate:

```
[root@bright82]# openssl x509 -in peterfile.pem -text -noout
```

© Bright Computing, Inc.
Data:

... Not After : Sep 21 13:18:27 2014 GMT
Subject: ... CN=democert

... Public-Key: (2048 bit)

X509v3 extensions:
1.3.6.1.4.4324.1:
  ..readonly
1.3.6.1.4.4324.2:
  ..peter

[root@bright82 #]

However, using the `openssl` utility for managing certificates is rather inconvenient. Bright Cluster Manager provides more convenient ways to do so, as described next.

**Listing Certificates**

All certificates that have been generated by the cluster are noted by CMDaemon.

**Listing certificates with cmsh:** Within the `cert` mode of `cmsh`, the `listcertificates` command lists all cluster certificates and their properties:

```
[root@bright82 ~]# cmsh -c "cert; listcertificates"
```

<table>
<thead>
<tr>
<th>Serial num</th>
<th>Days left</th>
<th>Profile</th>
<th>Country</th>
<th>Name</th>
<th>Revoked</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>36451</td>
<td>admin</td>
<td>US</td>
<td>Administrator</td>
<td>Yes</td>
</tr>
<tr>
<td>10</td>
<td>9</td>
<td>readonly ef</td>
<td>democert</td>
<td></td>
<td>Yes</td>
</tr>
<tr>
<td>11</td>
<td>36496</td>
<td>node</td>
<td>NL</td>
<td>52-54-00-de-e3-6b</td>
<td>No</td>
</tr>
<tr>
<td>12</td>
<td>36496</td>
<td>node</td>
<td>NL</td>
<td>52-54-00-44-fb-85</td>
<td>No</td>
</tr>
<tr>
<td>13</td>
<td>36496</td>
<td>admin</td>
<td>UK</td>
<td>otheradmin</td>
<td>No</td>
</tr>
</tbody>
</table>

**Listing certificates with Bright View:** The Bright View equivalent for listing certificates is via the click-path `Identity Management→Certificates` (figure 6.4):

![Figure 6.4: Bright View Certificates List Window](image)

In the certificates list, node certificates that are generated by the node-installer (section 5.4.1) for each node for CMDaemon use are listed.

© Bright Computing, Inc.
Custom certificates are also listed in the certificate lists.

Creating A Custom Certificate

Unlike node certificates, which are normally system-generated, custom certificates are typically generated by a user with the appropriate tokens in their profile, such as root with the admin profile. Such a user can create a certificate containing a specified profile, as discussed in the next section, by using:

- `cmsh`: with the `createcertificate` operation from within `cert` mode
- Bright View: via the clickpath `Identity Management → Users → Edit → Profile` to set the Profile.

Creating a new certificate for `cmsh` users: Creating a new certificate in `cmsh` is done from `cert` mode using the `createcertificate` command, which has the following help text:

```
[bright82->cert]% help createcertificate
Name: createcertificate - Create a new certificate
Usage: createcertificate <key-length> <common-name> <organization> <organizational-unit> <locality> <state> <country> <profile> <sys-login> <days> <key-file> <cert-file>
Arguments:
  key-file
    Path to key file that will be generated
  cert-file
    Path to pem file that will be generated
```

Accordingly, as an example, a certificate file with a read-only profile set to expire in 30 days, to be run with the privileges of user `peter`, can be created with:

**Example**

```
createcertificate 2048 democert a b c d ef readonly peter 30 /home/peter\ /peterfile.key /home/peter/peterfile.pem
```

Fri Aug 22 06:18:27 2014 [notice] bright82: New certificate request with ID: 1

```
[bright82->cert]% createcertificate 2048 democert a b c d ef readonly pe\ ter 30 /home/peter/peterfile.key /home/peter/peterfile.pem
Certificate key written to file: /home/peter/peterfile.key
Certificate pem written to file: /home/peter/peterfile.pem
```

The certificates are owned by the owner generating them, so they are root-owned if root was running cmsh. This means that user `peter` cannot use them until their ownership is changed to user `peter`:

**Example**

```
[root@bright82 ~]# cd /home/peter
[root@bright82 peter]# ls -l peterfile.*
-rw------- 1 root root 1704 Aug 22 06:18 peterfile.key
-rw------- 1 root root 1107 Aug 22 06:18 peterfile.pem
[root@bright82 peter]# chown peter:peter peterfile.*
```

Other users must have the certificate ownership changed to their own user names. Users associated with such a certificate can then carry out `cmdaemon` tasks that have a read-only profile, and `CMDaemon` sees such users as being user `peter`. Two ways of being associated with the certificate are:

© Bright Computing, Inc.
1. The paths to the pem and key files can be set with the -i and -k options respectively of cmsh. For example, in the home directory of peter, for the files generated in the preceding session, cmsh can be launched with these keys with:

```
[peter@bright82 ~] cmsh -i peterfile.pem -k peterfile.key
[bright82]% quit
```

2. If the -i and -k options are not used, then cmsh searches for default keys. The default keys for cmsh are under these paths under $HOME, in the following order of priority:

   (a) .cm/admin.{pem,key}
   (b) .cm/cert.{pem,key}

Creating a custom certificate for Bright View users: As in the case of cmsh, a Bright View user having a sufficiently privileged tokens profile, such as the admin profile, can create a certificate and key file for themselves or another user. This is done by associating a value for the Profile from the Add or Edit dialog for the user (figure 6.2).

The certificate files, cert.pem and cert.key, are then automatically placed in the following paths and names, under $HOME for the user:

- .cm/admin.{pem,key}
- .cm/cert.{pem,key}

Users that authenticate with their user name and password when running Bright View use this certificate for their Bright View clients, and are then restricted to the set of tasks allowed by their associated profile.

6.4.3 Creation Of Custom Certificates With Profiles, For Users Managed By An External LDAP

The use of an external LDAP server instead of Bright Cluster Manager’s for user management is described in section 6.3. Generating a certificate for an external LDAP user must be done explicitly in Bright Cluster Manager. This can be carried out with the external-user-cert script, which is provided with the cluster-tools package. The package is installed by default with Bright Cluster Manager.

Running the external-user-cert script embeds the user and profile in the certificate during certificate generation. The script has the following usage:

```
external-user-cert -h
```

Usage: for a single profile: external-user-cert <profile> <user> [<user> ... ]

```
--home=<home-prefix> [-g <group>] [-o]
```

for several profiles: external-user-cert --home=<home-prefix> --file=<inputfile> [-g <group>]

where lines of <inputfile> have the syntax

```
<profile> <user> [<user> ... ]
```

Options:

- `-h, --help`
  show this help message and exit

- `-file=FILE`
  input FILE

- `-home=HOME_PATH`
  path for home directories, default /home/

- `-g GROUP`
  name of primary group, e.g. wheel

- `-o`
  overwrite existing certificates

Here,

- `<profile>` should be a valid profile

© Bright Computing, Inc.
One or more external LDAP user certificates can be created by the script. The certificate files generated are `cert.pem` and `cert.key`. They are stored in the home directory of the user.

For example, a user spongebob that is managed on the external server, can have a read-only certificate generated with:

```bash
# external-user-cert readonly spongebob --home=/home
```

If the home directory of spongebob is `/home/spongebob`, then the key files that are generated are `/home/spongebob/.cm/cert.key` and `/home/spongebob/.cm/cert.pem`.

Assuming no other keys are used by cmsh, a cmsh session that runs as user spongebob with read-only privileges can now be launched:

```
$ module load cmsh
$ cmsh
```

If other keys do exist, then they may be used according to the logic explained in item 2 on page 234.

### 6.4.4 Logging The Actions Of CMDaemon Users

The following directives allow control over the logging of CMDaemon user actions.

- **CMDaemonAudit**: Enables logging
- **CMDaemonAuditorFile**: Sets log location
- **DisableAuditorForProfiles**: Disables logging for particular profiles

Details on these directives are given in Appendix C.
Workload Management

For clusters that have many users and a significant load, a workload management system allows a more efficient use of resources to be enforced for all users than if there were no such system in place. This is because without resource management, there is a tendency for each individual user to over-exploit common resources.

When a workload manager is used, the user submits a batch (i.e. non-interactive) job to it. The workload manager assigns resources to the job, and checks the current availability as well as checking its estimates of the future availability of the cluster resources that the job is asking for. The workload manager then schedules and executes the job based on the assignment criteria that the administrator has set for the workload management system. After the job has finished executing, the job output is delivered back to the user.

Among the hardware resources that can be used for a job are GPUs. Installing CUDA software to enable the use of GPUs is described in section 7.4 of the Installation Manual. Configuring GPU settings for Bright Cluster Manager is described in section 3.13.3 of the Administration Manual. Configuring GPU settings for an individual workload manager is described in the section on getting that particular workload manager up and running.

The details of job submission from a user’s perspective are covered in the User Manual. Sections 7.1–7.5 cover the installation procedure to get a workload manager up and running. Sections 7.6–7.7 describe how Bright View and cmsh are used to view and handle jobs, queues and node drainage. Section 7.8 shows examples of workload manager assignments handled by Bright Cluster Manager. Section 7.9 describes the power saving features of workload managers. Section 7.10 describes cgroups, a resources limiter, mostly in the context of workload managers. Section 7.11 describes custom workload manager customizations for settings other than the common settings covered by Bright Cluster Manager.

7.1 Workload Managers Choices

Some workload manager packages are installed by default, others require registration from the distributor before installation.

During cluster installation, a workload manager can be chosen (figure 3.26 of the Installation Manual) for setting up. The choices are:

- None
- Slurm v18.08.8 (default)
- Grid Engine 2011.11p1. An open source development of Sun Grid Engine (SGE)

A flavor of SGE is Univa Grid Engine (UGE) (section 7.5.2) developed by Univa. Bright Cluster Manager 8.2 supports integration with UGE versions 8.2.0 and higher at the time of writing of this section (October 2017).
• PBS Pro CE v18.1.2
• PBS Pro v18.2.2

These workload managers can also be chosen and set up later using the `wlm-setup` tool (section 7.3). Besides the preceding workload managers, the installation of the following workload managers is also possible, and described in their own sections:

• Load Sharing Facility (LSF) v9, v10 (section 7.5.5)

7.2 Forcing Jobs To Run In A Workload Management System

Another preliminary step is to consider forcing users to run jobs only within the workload management system. Having jobs run via a workload manager is normally a best practice.

For convenience, the Bright Cluster Manager defaults to allowing users to login via `ssh` to a node, using the authorized keys files stored in each users directory in `/home` (section 2.3.2). This allows users to run their processes outside the workload management system without restriction. For clusters with a significant load this policy results in a sub-optimal use of resources, since such unplanned-for jobs disturb any already-running jobs.

Disallowing user logins to nodes, so that users have to run their jobs through the workload management system, means that jobs are then distributed to the nodes only according to the planning of the workload manager. If planning is based on sensible assignment criteria, then resources use is optimized—which is the entire aim of a workload management system in the first place.

7.2.1 Disallowing User Logins To Regular Nodes Via `cmsh`

The `usernodelogin` setting of `cmsh` restricts direct user logins from outside the workload manager, and is thus one way of preventing the user from using node resources in an unaccountable manner. The `usernodelogin` setting is applicable to node categories only, rather than to individual nodes.

In `cmsh` the attribute of `usernodelogin` is set from within category mode:

**Example**

```
[root@bright82 ~]# cmsh
[bright82]% category use default
[bright82->category[default]]% set usernodelogin onlywhenjob
[bright82->category*[default*]]% commit
```

The attributes for `usernodelogin` are:

• **always** (the default): This allows all users to `ssh` directly into a node at any time.

• **never**: This allows no user other than root to directly `ssh` into the node.

• **onlywhenjob**: This allows the user to `ssh` directly into the node when a job is running on it. It typically also prevents other users from doing a direct `ssh` into the same node during the job run, since typically the workload manager is set up so that only one job runs per node. However, an `ssh` session that is already running is not automatically terminated after the job is done.

  – Some cluster administrators may wish to allow some special user accounts to override the `onlywhenjob` setting, for example for diagnostic purposes. Before giving the details of how to override the setting, some background explanation is probably useful:

  The `onlywhenjob` setting works with the PAM system, and adds the following line to `/etc/pam.d/sshd` on the regular nodes:

  ```
  account required pam_bright.so
  ```

© Bright Computing, Inc.
Nodes with the `onlywhenjob` restriction can be configured to allow a particular set of users to access them despite the restriction by whitelisting them in the PAM system, as follows: Within the software image `<node image>` used by the node, that is under `/cm/images/<node image>`, the administrator can add the set of user accounts to the file `etc/security/pam_bright.d/pam_whitelist.conf`. This file is installed in the software image with a chroot installation (section 11.4) of the `cm-libpam` package.

Other adjustments to PAM configuration, such as the number of attempted logins and the associated wait time per login, can be carried by editing the `/etc/security/pam_bright.d/cm-check-alloc.conf` file.

The image can then be updated to implement the whitelist, by running the `imageupdate` command in `cmsh` (section 5.6.2), or by clicking the `Update node` option in Bright View (section 5.6.3).

### 7.2.2 Disallowing User Logins To Regular Nodes Via Bright View

In Bright View, user node login access is set via a category setting, for example for the default category via the clickpath:

**Grouping** → **Node categories[default]** → **Edit** → **Settings** → **User node login** (figure 7.1):

![Figure 7.1: Disallowing User Logins To Nodes Via Bright View](image)

### 7.2.3 Disallowing Other User Processes Outside Of Workload Manager User Processes

Besides disabling user logins, administrators may choose to disable interactive jobs in the workload management system as an additional measure to prevent users from starting jobs on other nodes.

Administrators may also choose to set up scripts that run after job execution. Such scripts can terminate user processes outside the workload manager, as part of a policy, or for general administrative hygiene. These are Epilog scripts and are part of the workload manager.

The workload management system documentation has more on configuring these options.

### 7.3 Installation Of Workload Managers

Normally the administrator selects a workload manager to be used during Bright Cluster Manager installation (figure 3.26 of the *Installation Manual*). A workload manager may however also be added and configured after Bright Cluster Manager has been installed, using `wlm-setup`, which has the following usage:

```
[root@bright82 ~]# wlm-setup -h
```

© Bright Computing, Inc.
7.3.1 Setting Up, Enabling, And Disabling The Workload Manager With `wlm-setup`
The following options from `wlm-setup` must be chosen to set up, enable or disable a workload manager for Bright Cluster Manager:

- `-w` or `--wlmanager`: the **workload manager** option, together with `<name>`, the workload manager name. The value of `<name>` can be one of
  - `slurm`
  - `sge`
  - `uge` (This has a prerequisite that the packages must be installed from the UGE site first, as explained in section 7.5.3)
  - `pbspro` (PBS Pro commercial edition)
  - `pbspro-ce` (PBS Pro Community Edition)
  - `lsf`

and for the named workload manager, either:

- `-s` or `--setup`: the **setup** option. This does a basic setup of the workload manager, initializing the database for the workload manager, setting up default queues, and enabling the workload manager clients for the default category. It can optionally take the further options of section 7.3.2 or

- `-e` or `--enable`: the **enable** option. Enabling assigns workload manager client roles and default queues to the applicable node categories. This option need not be run if the `setup` option has just run. CMDaemon attempts to run workload managers that have been enabled.

- `-p` or `--powersave` option will also allow power management, for `slurm` only, for Slurm clients only, for non-head nodes only. Independently of `wlm-setup`, power management can also be allowed by using Slurm roles (section 7.9.1) without using `wlm-setup`. When using Slurm roles, the `powersavingallowed` parameter in the `slurmclient` role allows power management for head nodes too if set for a head node, in the unlikely case that the administrator finds it is acceptable to power off a head node that is not running Slurm jobs.

If power saving is allowed for a node, then the `powersavingenabled` parameter in the `slurmsServer` role is what actually enables power management for that node.

or

- `-d` or `--disable`: the **disable** option, which disables the workload manager

- `-p` or `--powersave` option will disallow power management only, for `slurm` only, for Slurm clients only.

or

- `-i` or `--image`: the **image** option, which places the job execution daemons in software images, for example when a new node category is added to the cluster. The default image directory is `/cm/images/default-image`.

The `--setup` option of `wlm-setup` installs a workload manager package along with its scheduler. Roles, queues, and databases used by the workload manager are initialized on the head. Software images stored on the head node are also set up. However, the nodes themselves are only updated after a reboot, or manually running `imageupdate` (section 5.6.2)—the idea being to avoid an automatic update so that an administrator is encouraged to check via a dry-run if unintended consequences would happen.
The enable and disable options of `wlm-setup` are the same as those that take place during role assignment (sections 7.4.1 and 7.4.2). These options therefore make CMDaemon enable or disable the workload manager as a service.

For example, setting up SGE can be done as follows:

**Example**

```
[root@bright82 ~]# wlm-setup -w sge -s

Disabling sge services ...... [ OK ]
Initializing sge setup ...... [ OK ]
Installing sge qmaster ...... [ OK ]
Updating image services ...... [ OK ]
Updating qmaster config ...... [ OK ]
Creating default sge setup ...... [ OK ]
Setting permissions ...... [ OK ]
Enabling sge services ...... [ OK ]
```

For example, Slurm job daemons can be installed in the node image `new-image` as follows:

**Example**

```
wlm-setup -w slurm -i -u /cm/images/new-image
```

If there are provisioning nodes, the `updateprovisioners` command (section 5.2.4) should be run after the software image is changed. The nodes can then simply be rebooted to pick up the new image, or alternatively, to avoid rebooting, the `imageupdate` command (section 5.6.2) places a new image on the node from the provisioner.

### 7.3.2 Other Options With `wlm-setup`

A full list of options to `wlm-setup` can be seen by running it with the `-h|--help` option. The list shows other options besides the ones listed in section 7.3.1. These other options are run as further options to the `-s|--setup` option, and are:

- **included images**  
  The included images option (`-u` or `--updateimages <[image1[,image2,...]>`) includes the comma-separated list of images

- **excluded images**  
  The excluded images option (`-x` or `--excludeimages <[image1[,image2,...]>`) excludes the comma-separated list of images

- **offload**  
  The offload option (`-o` or `--offload`) deals with setting up, enabling or disabling a workload manager server running on another specified node, other than the default head node:

  **Example**

  ```
wlm-setup -w slurm -s -o node003
  ````

- **head as a compute node**  
  This option (`-m` or `--mcompute`) sets the head node to join in as a compute node for job execution tasks in a workload management system. This can be significantly worthwhile on smaller clusters:

  **Example**

  ```
wlm-setup -w slurm -s -m
  ```

- **slots**  
  The slots option (`-n` or `--slots`) sets the maximum allowed value of slots, and is typically set to the number of cores per node:
Example

`wlm-setup -w slurm -s -n 4`

For workload managers, slots is the number of logical CPUs set per node.

- It is not recommended to set the number of slots to be greater than the number of cores for non-threading applications. That is because in such a case, when all the slots are running jobs, the maximum speed at which the jobs can run is usually reduced in comparison with the one-slot-per-core case, and the efficiency at which they run is also less.

- However increasing the slots value to be greater than the number of cores may improve performance for applications that are optimized to use threading, if they are running on cores that have hyperthreading enabled. Because enabling hyperthreading can increase power consumption and can result in unexpected performance issues, benchmarking is advised.

The number of slots can also be adjusted outside of `wlm-setup`, after setup, by using Bright Cluster Manager to set it in the workload manager client role.

**SGE and slots:** In SGE, `wlm-setup` sets the number of slots in a complex (`man (5) complex`) configuration file.

In practice, for SGE this means that the system administrator manually sets the number of slots to the number of cores per node during the initial Bright Cluster Manager installation, or during `wlm-setup`, or using `cmsh` or Bright View, just like in the other workload managers. The complex configuration file value is set automatically according to this value.

If an end user explicitly requests slots for a job `myjob` with:

**Example**

`qsub -l slots=2 myjob`

then SGE tries to allocate a node with 2 free slots. If there are no nodes with more than one slot, then `myjob` will not run.

The difference in comparison with the other workload managers occurs when a user needs to use a parallel engine driver such as Open MPI. In that case, the requested number of slots in SGE can be used by MPI processes as well as regular non-MPI processes. Thus, the user submission request:

**Example**

`qsub -l slots=2 -pe openmpi 32 myjob`

means that enough nodes with 2 free slots each should be made available to run all the 32 MPI processes being requested. Again, if there are no nodes with more than one slot, the job will not run. If `slots=2` is not specified in the `qsub` line, then the MPI processes are spread across whatever cores are available, if there are nodes with one or more free slots.

- **archives** The archives option (`-a` or `--archives-location`) sets the directory path to the archives files used for software installation. This is valid for UGE or LSF only.

**Example**

`wlm-setup -w lsf -s -a /root/lsf`
• **categories**  The categories option (-c or --categories) assigns the client role to a comma-separated list of categories instead of to the default category. The default category takes the value default by default. The default value for a category can be changed to another existing category by setting the value for the `defaultcategory` attribute for the base object in partition mode:

**Example**

```
[bright82]% partition use base
[bright82->partition[base]]% get defaultcategory
default
[bright82->partition[base]]% set defaultcategory fast; commit
```

**Example**

```
wm-setup -w slurm -s -c fast,superfast,ludicrous
```

• **license**  The license option (-l or --license <host:port,...>) sets the host and port pairings up for use with a dedicated license server for PBSPro.

### 7.3.3 Prolog And Epilog Scripts

#### What Prolog And Epilog Scripts Do

The workload manager runs prolog scripts before job execution, and epilog scripts after job execution. The purpose of these scripts can include:

- checking if a node is ready before submitting a job execution that may use it
- preparing a node in some way to handle the job execution
- cleaning up resources after job execution has ended.

The administrator can run custom prolog or epilog scripts for the queues from CMDaemon for SGE, or LSF, by setting such scripts in the Bright View or cmsh front ends. Default epilogs and prologs are created automatically when a new queue is created.

**Example**

```
[bright82->jobqueue]% add lsf newq
[bright82->jobqueue*(lsf)->newq*] show | grep . | grep -i epilog
Epilog /cm/local/apps/cmd/scripts/epilog
Prolog/Epilog user root
```

For PBS Pro and Slurm, there are global prolog and epilog scripts, but editing them is not recommended. Indeed, in order to discourage editing them, the scripts cannot be set via the cluster manager front ends. Instead the scripts must be placed by the administrator in the software image, and the relevant nodes updated from the image.

#### Detailed Workings Of Prolog And Epilog Scripts

Even though it is not recommended, some administrators may nonetheless wish to link and edit the scripts directly for their own needs, outside of the Bright View or cmsh front ends. A more detailed explanation of how the prolog scripts work therefore follows:

The prolog scripts have names and function according to their locations. The scripts are placed in two directories:

1. In the **main scripts directory**, at:
In this directory, a main prolog script, `prolog`, can call a sequence of rc.d-style prolog scripts for a particular workload manager in an rc.d-style directory.

2. **rc.d-style prolog directory**, at:

```
/cm/local/apps/<workload manager>/var/prologs/
```

In this directory, prolog scripts, if they exist, are stored in a directory path associated with the workload manager name. The names of the scripts have suffixes and prefixes associated with them that make them run in special ways, as follows:

- **suffixes used in the rc.d-style directory:**
  - `-prejob`: script runs prior to all jobs
  - `--cmsub`: script runs prior to job run in a cloud

- **prefixes used in the rc.d-style directory:**
  - `00-` to `99-`

  Number prefixes determine the order of script execution. This is like for SysV-style rc.d names, where scripts with the lower number are run earlier. Hence the terminology “rc.d-style” associated with these prolog scripts.

The script names can therefore look like:

**Example**

- `00-prolog-prejob`
- `10-prolog-cmsub`

Return values for the rc.d-style scripts have these meanings:

- **0**: the next script in the directory is run.
- **A non-zero return value**: no further scripts are executed from the rc.d-style directory.

Often, the script in an rc.d-style prolog directory is not a real script but a symlink, with the symlink going to a general script located in the main scripts directory. In that case, this general script is then able to take care of what is expected of the symlink. The name of the symlink, and destination file, usually hints at what the script is expected to do.

For example, the Pbs Pro workload manager uses the symlink `10-prolog-prejob` within the rc.d-style directory `/cm/local/apps/pbspro/var/prologs/`. The symlink links to the script `prolog-prejob` within the main scripts directory `/cm/local/apps/cmd/scripts/`. In this case, the script is expected to run prior to the job.

Epilog scripts, which run after a job run, can be used for SGE and Slurm.

The PBS Pro and Slurm prolog and epilog scripts are global in effect.
7.4 Enabling, Disabling, And Monitoring Workload Managers

Workload Manager Package Configuration For Prolog And Epilog Scripts

Each workload manager package configures prolog- and epilog-related scripts or links during installation, as follows:

- **Slurm**
  - `prolog-prejob`: in the main scripts directory, is assigned to `PrologSlurmctld` in `/etc/slurm/slurm.conf`. It runs by default during job execution, and is executed with slurm user permissions.
  - `prolog`: in the main scripts directory, is assigned to the variable `Prolog` in `/etc/slurm/slurm.conf`. The script executes the `<number>-prolog{-cmsub|-prejob}` scripts located in the rc.d-style directory, if they exist. By default, none exist. The `epilog` script in the main scripts directory follows the same pattern, with the appropriate name changes.

- **SGE, LSF**
  - `prolog`: in the main scripts directory, executes any `<number>-prolog{-cmsub|-prejob}` scripts located in the rc.d-style directory, if they exist.
    It is set by default to execute `10-prolog-prejob` in the rc.d-style directory. This in turn is configured as a symlink to `prolog-prejob` in the main scripts directory, which is able to handle the execution of scripts in a general manner. Any further scripts in the rc.d-style directories, whether for prologs or epilogs, are then also processed.

- **PBS Pro**
  - `mom_priv/prologue` (with the -ue ending) in the rc.d-style directory, is a symlink to `prolog` in the main scripts directory. This in turn executes any `<number>-prolog{-cmsub|-prejob}` scripts located in the rc.d-style directory, if they exist. Similar to this are the `epilogue` scripts in the rc.d-style directory.

7.4 Enabling, Disabling, And Monitoring Workload Managers

After a workload manager package is installed and initialized with `wlm-setup` (section 7.3), it can also be enabled or (if already enabled) disabled, with `wlm-setup`. Enabling and disabling means the workload management services start or stop.

Alternatively, a workload manager can be enabled or disabled by the administrator with Bright View or `cmsh`. This is described further on in this section.

In Bright Cluster Manager 8.2, workload managers can even run concurrently. For example, Slurm, SGE, and PBS Pro run at the same time in the cluster, for example with one workload manager assigned to one category. However, this can easily lead to confusion. Running only one workload manager is a more sane practice, and is strongly urged for production environments.

From the Bright View or `cmsh` point of view a workload manager consists of

- a workload manager server, usually on the head node
- workload manager clients, usually on the compute nodes

For the administrator, enabling or disabling the servers or clients is then simply a matter of assigning or unassigning a particular workload manager server or client role on the head or compute nodes, as deemed appropriate.

The administrator typically also sets up an appropriate workload manager environment module (`slurm`, `sge`, `uge`, or `pbspro`), so that it is loaded up for the end user (section 2.2.3).
7.4.1 Enabling And Disabling A Workload Manager With Bright View

A particular workload manager package may be set up, but the workload manager may not be enabled. This can happen, for example, if using `wlm-setup` (section 7.3) to install the package without enabling it. It may also happen, for example, if disabling a workload manager that was previously enabled.

The workload manager client and server can be enabled from Bright View by assigning the appropriate role to it. Within the role, the properties of the workload manager may be further configured.

Workload Manager Role Assignment To An Individual Node With Bright View

Workload Manager Server  A server or client, for example, Slurm server, can be assigned to a node, for example `bright82`, via the clickpath:

`Devices → Head Nodes[bright82] → Edit → Settings → Roles → Slurm server role`

The large number of roles means that using the filter to narrow down the search, using text from role, is usually convenient (figure 7.2).

![Figure 7.2: Workload Management Role Assignment On A Head Node](image)

Options can be set within the workload manager, within the role. For example, for Slurm, a `builtin` or `backfill` option can be set for the `Scheduler` parameter. The workload manager server role is then saved with the selected options (figure 7.3). For starting it up on non-head nodes (but not for a head node), the `imageupdate` command (section 5.6.2) is then run. The workload manager server process and any associated schedulers then automatically start up.
Enabling, Disabling, And Monitoring Workload Managers

Similarly, the workload manager client process can be enabled on a node or head node by having the workload manager client role assigned to it. Some basic options can be set for the client role right away.

Saving the role, and then running `imageupdate` (section 5.6.2), automatically starts up the client process with the options chosen, and managed by CMDaemon.

Workload Manager Role Assignment To A Category With Bright View

While workload manager role assignment can be done as described in the preceding text for individual non-head nodes, it is usually more efficient to assign roles using categories due to the large number of compute nodes in typical clusters.

All non-head nodes are by default placed in the `default` category. This means that by default roles in the category are automatically assigned to all non-head nodes, unless by way of exception an individual node configuration overrides the category setting and uses its own role setting instead.

Viewing the possible workload manager roles for the category `default` is done by using the click-path:

* Grouping → Node categories [default] → Edit → Settings → Roles → Add

Using the filter to narrow down the search is usually convenient. Once the role is selected its options can be edited and saved.

For compute nodes, the role assigned is usually a workload manager client. The assigned role for a compute node allows queues and GPUs to be specified, and other parameters depending on the workload manager used.

The workload manager server role can also be assigned to a non-head node. For example, a Slurm server role can be carried out by a non-head node. This is the equivalent to the `offload` option of `wlm-setup`.

Saving the roles with their options and then running `imageupdate` (section 5.6.2) automatically starts up the newly-configured workload manager.

Workload Manager Role Options With Bright View

Each compute node role (workload manager client role) has options that can be set for GPUs, Queues, and Slots.

- **Slots**, in a workload manager, corresponds in Bright Cluster Manager to:
  - the CPUs setting (a `NodeName` parameter) in Slurm’s `slurm.conf`
  - the `np` setting in PBS Pro,
and is normally set to the number of cores per node.

In LSF setting the number of slots for the client role to 0 means that the client node does not run jobs on itself, but becomes a submit host, which means it is able to forward jobs to other client nodes.

- Queues with a specified name are available in their associated role after they are created. The creation of queues is described in sections 7.6.2 (using Bright View) and 7.7.2 (using cmsh).

Among the other options displayed are installation path or spool path settings.

All server roles also provide the option to enable or disable the External Server setting. Enabling it means that the server is no longer managed by Bright Cluster Manager, but provided by an external device.

An external Torque server role on an enabled node also means that additional scheduler services—Maui or Moab—are not started, nor are they monitored, on that node.

The cmsh equivalent of this is described on page 250.

7.4.2 Enabling And Disabling A Workload Manager With cmsh

A particular workload manager package may be set up, but not enabled. This can happen, for example, if using wlm-setup (section 7.3) on the package without enabling it. The workload manager client and server can be enabled from cmsh by assigning it from within the roles submode. Within the assigned role, the properties of the workload manager may be further configured.

Workload Manager Role Assignment To A Category With cmsh

Workload manager role assignment of a node category is done using category mode, using the category name, and assigning a role from the roles submode:

Example

```
[root@bright82 ~]# cmsh
[bright82]% category
[bright82->category]% use default
[bright82->category[default]]% roles
[bright82->category[default]->roles]% assign slurmclient
[bright82->category[default]->roles[slurmclient]]% commit
[bright82->category[default]->roles[slurmclient]]%
```

After workload manager roles are assigned or unassigned, and after running imageupdate (section 5.6.2) for non-head nodes, the associated workload manager services automatically start up or stop as appropriate.

Workload Manager Role Assignment To An Individual Node With cmsh

In cmsh, assigning a workload manager role to a head node is done in device mode, using master as the device, and assigning the workload manager role from the roles submode:

Example

```
[root@bright82 ~]# cmsh
[bright82]% device
[bright82->device]% use bright82
[bright82->device[bright82]]% roles
[bright82->device[bright82]->roles]% assign slurmserver
[bright82->device[bright82]->roles[slurmserver]]% commit
[bright82->device[bright82]->roles[slurmserver]]%
```

For regular nodes, role assignment is done via device mode, using the node name, and assigning a role from the roles submode:

© Bright Computing, Inc.
Example

[root@bright82 ~]# cmsh
[bright82]# device
[bright82->device]# use node001
[bright82->device[node001]]# roles
[bright82->device[node001]->roles]# assign slurmclient
[bright82->device[node001]->roles*|slurmclient*]# commit
[bright82->device[node001]->roles[slurmclient]]%

Role assignment values set in device mode have precedence over any role assignment values set in category mode for that node. This means, for example, that if a node is originally in a node category with a slurmclient role and queues set, then when the node is assigned a slurmclient role from device mode, its queue properties are empty by default.

Setting Options For Workload Manager Settings With cmsh

In the preceding text, it is explained how the workload manager client or server is assigned a role (such as slurmclient or slurmserver) within the roles submode. It is done from within a main mode of category or devices.

Options for workload managers in general: Whatever the main mode used, the workload manager settings can then be handled with the usual object commands introduced in section 2.5.3.

For example, the number of slots can be set for Slurm clients as follows:

Example

[bright82->category[default]->roles[slurmclient]]# show
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add services</td>
<td>yes</td>
</tr>
<tr>
<td>Name</td>
<td>slurmclient</td>
</tr>
<tr>
<td>Provisioning associations</td>
<td>&lt;0 internally used&gt;</td>
</tr>
<tr>
<td>Revision</td>
<td></td>
</tr>
<tr>
<td>Type</td>
<td>SlurmClientRole</td>
</tr>
<tr>
<td>Slots</td>
<td>0</td>
</tr>
<tr>
<td>Queues</td>
<td></td>
</tr>
<tr>
<td>All Queues</td>
<td>no</td>
</tr>
<tr>
<td>GPUs</td>
<td>0</td>
</tr>
<tr>
<td>Weight</td>
<td>0</td>
</tr>
<tr>
<td>Boards</td>
<td>0</td>
</tr>
<tr>
<td>SocketsPerBoard</td>
<td>0</td>
</tr>
<tr>
<td>Power Saving Allowed</td>
<td>no</td>
</tr>
<tr>
<td>Reason</td>
<td></td>
</tr>
<tr>
<td>NodeAddr</td>
<td></td>
</tr>
<tr>
<td>Cores Per Socket</td>
<td>0</td>
</tr>
<tr>
<td>Features</td>
<td></td>
</tr>
<tr>
<td>Port</td>
<td>0</td>
</tr>
<tr>
<td>RealMemory</td>
<td>0</td>
</tr>
<tr>
<td>Sockets</td>
<td>0</td>
</tr>
<tr>
<td>ThreadsPerCore</td>
<td>0</td>
</tr>
<tr>
<td>TmpDisk</td>
<td>0</td>
</tr>
<tr>
<td>CPU Spec List</td>
<td></td>
</tr>
<tr>
<td>Core Spec Count</td>
<td>0</td>
</tr>
<tr>
<td>Mem Spec Limit</td>
<td>0</td>
</tr>
<tr>
<td>Node Customizations</td>
<td>&lt;0 in submode&gt;</td>
</tr>
<tr>
<td>Generic Resources</td>
<td>&lt;0 in submode&gt;</td>
</tr>
</tbody>
</table>
Similarly, Slurm server options can be accessed from an assigned slurmserver role:

Example

```
[bright82->device[bright82]->roles[slurmserver]]% use slurmserver
[bright82->device[bright82]->roles[slurmserver]]% show
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add services</td>
<td>yes</td>
</tr>
<tr>
<td>Cgroups</td>
<td>&lt;submode&gt;</td>
</tr>
<tr>
<td>External Server</td>
<td>no</td>
</tr>
<tr>
<td>GresTypes</td>
<td>gpu</td>
</tr>
<tr>
<td>Name</td>
<td>slurmserver</td>
</tr>
<tr>
<td>Placeholders</td>
<td>&lt;0 in submode&gt;</td>
</tr>
<tr>
<td>Provisioning associations</td>
<td>&lt;0 internally used&gt;</td>
</tr>
<tr>
<td>Revision</td>
<td></td>
</tr>
<tr>
<td>Scheduler</td>
<td>backfill</td>
</tr>
<tr>
<td>Type</td>
<td>SlurmServerRole</td>
</tr>
<tr>
<td>Fast Schedule</td>
<td>0</td>
</tr>
<tr>
<td>Power Saving Enabled</td>
<td>no</td>
</tr>
<tr>
<td>Resume Program</td>
<td>/cm/local/apps/cluster-tools/wlm/scripts/slurmpoweron</td>
</tr>
<tr>
<td>Resume Timeout</td>
<td>60</td>
</tr>
<tr>
<td>Suspend Program</td>
<td>/cm/local/apps/cluster-tools/wlm/scripts/slurmpoweroff</td>
</tr>
<tr>
<td>Suspend Time</td>
<td>300</td>
</tr>
<tr>
<td>Suspend Timeout</td>
<td>30</td>
</tr>
<tr>
<td>Etc</td>
<td>/cm/shared/apps/slurm/var/etc</td>
</tr>
<tr>
<td>Prefix</td>
<td>/cm/shared/apps/slurm/current</td>
</tr>
<tr>
<td>Epilog Program</td>
<td>/cm/local/apps/cmd/scripts/epilog</td>
</tr>
<tr>
<td>Prolog Program</td>
<td>/cm/local/apps/cmd/scripts/prolog</td>
</tr>
<tr>
<td>Prolog Slurmctld Program</td>
<td>/cm/local/apps/cmd/scripts/prolog-prejob</td>
</tr>
</tbody>
</table>

```
[bright82->device[bright82]->roles[slurmserver]]%
```

**Option to set an external workload manager:** A workload manager can be set to run as an external server from within a device mode role:

Example

```
[bright82->device[bright82]->roles[slurmserver]]% set externalserver on
[bright82->device[bright82]->roles[slurmserver*]]% commit
```

For convenience, setting it on the head node is recommended.

The Bright View equivalent of configuring externalserver is described on page 248.

### 7.4.3 Monitoring The Workload Manager Services

By default, the workload manager services are monitored. The Bright Cluster Manager attempts to restart the services using the service tools (section 3.11), unless the role for that workload manager service is disabled, or the service has been stopped (using `cmsh`). Workload manager roles and corresponding services can be disabled using `wlm-setup` (section 7.3.1), Bright View (section 7.4.1), or `cmsh` (section 7.4.2).

The daemon service states can be viewed for each node via the shell, `cmsh`, or Bright View (section 3.11).
Queue submission and scheduling daemons normally run on the head node. From Bright View their states are viewable via the clickpath to the services running on the node (figure 3.16), for example, for a head node:

Devices→Nodes[bright82]→Settings→Services

The job execution daemons run on compute nodes. Their states are viewable via a similar clickpath.

From cmsh the services states are viewable from within device mode, using the services command. One-liners from the shell to illustrate this are (output elided):

Example

```
[root@bright82 ~]# cmsh -c "device services node001; status"
  sgeexecd [ UP ]
[root@bright82 ~]# cmsh -c "device services bright82; status"
  ...  
    sge [ UP ]
```

### 7.5 Configuring And Running Individual Workload Managers

Bright Cluster Manager deals with the various choices of workload managers in as generic a way as possible. This means that not all features of a particular workload manager can be controlled, so that fine-tuning must be done through the workload manager configuration files. Workload manager configuration files that are controlled by Bright Cluster Manager should normally not be changed directly because Bright Cluster Manager overwrites them. However, overwriting by CMDaemon is prevented on setting the directive:

\[
\text{FreezeChangesTo<workload manager>Config = <true|false>}
\]

in `cmd.conf` (Appendix C), where `<workload manager>` takes the value of Slurm, SGE, LSF, or PBSPro, as appropriate. The value of the directive defaults to `false`.

A list of configuration files that are changed by CMDaemon, the items changed, and the events causing such a change are listed in Appendix H.

A very short guide to some specific workload manager commands that can be used outside of the Bright Cluster Manager 8.2 system is given in Appendix F.

#### 7.5.1 Configuring And Running Slurm

**Slurm Versions**

Bright Cluster Manager version 8.2, from release version 8.2-25 onwards, is integrated with Slurm by using either of the following packages:

- `slurm`. This is based on Slurm version 18, and is updated in the Bright Cluster Manager repositories for important updates from upstream. At the time of writing (July 2019) the package is based on Slurm release 18.08.8. This is the recommended version for production clusters, and is installed by default.

- `slurm19`. This is based on Slurm version 19.05, and is updated in the Bright Cluster Manager repositories for important updates from upstream.

- `slurm20`. This is based on Slurm version 20.02, and is updated in the Bright Cluster Manager repositories for important updates from upstream.

- `slurm20.11`. This is based on Slurm version 20.11. It should not be confused with version 20, which is a different major slurm version. The `slurm20.11` package is updated in the Bright Cluster Manager repositories for important updates from upstream. Packages for this version have names that start with `slurm20.11`, where 11 here is not a package version, but a part of the package name.
The packages for versions 19, 20 and 20.11 are provided because they may have new features that are absent in version 18.

The package `slurm-latest`, which in earlier Bright Cluster Manager 8.2 releases had up to Slurm version 18.08.7 as the latest release, is obsolete from version 8.2-11 onward. It should be replaced by removing the `slurm-latest` package and installing either `slurm` or `slurm19`.

It is left up to the cluster administrator to decide which version to use.

Bright Cluster Manager provides a similar level of workload manager integration for the supported Slurm packages.

If switching from one major Slurm version to another—for example, from `slurm` to `slurm19`—then the administrator must uninstall the old Slurm packages first, in order to avoid a conflict between the versions.

The following example demonstrates how to remove the existing `slurm` package and update to another `slurm19` for RHEL-based distributions:

**Example**

```
[root@bright82 ~]# rpm -qa | grep slurm | xargs -p rpm -e
[root@bright82 ~]# rpm -qa -r /cm/images/default-image |grep slurm |xargs -p rpm -r /cm/images/default-image -e
[root@bright82 ~]# yum install slurm19-client slurm19-slurmdbd slurm19-perlapli slurm19-contribs slurm19
[root@bright82 ~]# yum install --installroot=/cm/images/default-image slurm19-client
```

If either `slurm` or `slurm19` is installed, then the administrator can run `wlm-setup` using the workload manager name `slurm`—that is without the 19 suffix—to set up Slurm. The roles at node level, or category level—`slurmserver` and `slurmclient`—work with either Slurm version.

**Configuring Slurm**

After package setup is done with `wlm-setup` (section 7.3), Slurm software components are installed in `/cm/shared/apps/slurm/current`.

Slurm clients and servers can be configured to some extent via role assignment (sections 7.4.1 and 7.4.2).

Using `cmsh`, advanced option parameters can be set under the `slurmclient` role:

For example, the number of cores per socket can be set:

**Example**

```
[bright82->category[default]->roles[slurmclient]]% set corespersocket 2
[bright82->category[default]->roles[slurmclient]]% commit
```

In order to configure generic resources, the `genericresources` mode can be used to set a list of objects. Each object then represents one generic resource available on nodes.

Each value of `name` in `genericresources` must already be defined in the list of `GresTypes`. The list of `GresTypes` is defined in the `slurmserver` role.

Several generic resources entries can have the same value for `name` (for example `gpu`), but must have a unique alias. The alias is a string that is used to manage the resource entry in `cmsh` or in Bright View. The string is enclosed in square brackets in `cmsh`, and is used instead of the `name` for the object. The alias does not affect Slurm configuration.

For example, to add two GPUs for all the nodes in the default category which are of type `k20xm`, and to assign them to different CPU cores, the following `cmsh` commands can be run:

**Example**

```
[bright82]% category use default
[bright82->category[default]]% roles
[bright82->category[default]->roles]% use slurmclient
[...[slurmclient]]% genericresources
```
7.5 Configuring And Running Individual Workload Managers

```bash
[[slurmclient*]->genericresources[gpu0*]]% add gpu0
[[slurmclient*]->genericresources[gpu0*]]% set name gpu
[[slurmclient*]->genericresources[gpu0*]]% set file /dev/nvidia0
[[slurmclient*]->genericresources[gpu0*]]% set cores 0-7
[[slurmclient*]->genericresources[gpu0*]]% set type k20xm
[[slurmclient*]->genericresources[gpu0*]]% add gpu1
[[slurmclient*]->genericresources[gpu1*]]% set name gpu
[[slurmclient*]->genericresources[gpu1*]]% set file /dev/nvidia1
[[slurmclient*]->genericresources[gpu1*]]% set cores 8-15
[[slurmclient*]->genericresources[gpu1*]]% set type k20xm
[[slurmclient*]->genericresources[gpu1*]]% commit
[[slurmclient*]->genericresources[gpu1*]]% list

<table>
<thead>
<tr>
<th>Alias (key)</th>
<th>Name</th>
<th>Type</th>
<th>Count</th>
<th>File</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>gpu0</td>
<td>gpu</td>
<td></td>
<td>/dev/nvidia0</td>
</tr>
<tr>
<td></td>
<td>gpu1</td>
<td>gpu</td>
<td></td>
<td>/dev/nvidia1</td>
</tr>
</tbody>
</table>
```

After the generic resources are added, Bright Cluster Manager updates `/etc/slurm/gres.conf`, so that it looks like the following:

**Example**

```bash
[root@bright82 ~]# cat /etc/slurm/gres.conf
# This section of this file was automatically generated by cmd. Do not edit manually!
# BEGIN AUTOGENERATED SECTION -- DO NOT REMOVE
NodeName=node[001,002] Name=gpu Type=k20xm File=/dev/nvidia0 Cores=0-7
NodeName=node[001,002] Name=gpu Type=k20xm File=/dev/nvidia1 Cores=8-15
# END AUTOGENERATED SECTION -- DO NOT REMOVE
[root@bright82 ~]#
```

Since Bright Cluster Manager version 8.2 and higher, a single `gres.conf` configuration file, located at `/cm/shared/apps/slurm/var/etc/gres.conf` is used. A symlink to this path is created on each node of the cluster at `/etc/slurm/gres.conf`.

The parameter value for a Slurm option is used unless it has the value: 0.

A parameter value of 0 means that the default values of Slurm are used. These usually have the value: 1.

The advanced options that CMDaemon manages for Slurm are:
<table>
<thead>
<tr>
<th>Bright View Option</th>
<th>Slurm Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Features</td>
<td>Feature=&lt;string&gt; entry in the file slurm.conf</td>
<td>Arbitrary strings can be entered to indicate some characteristics of a node, one string per entry. For example: text1 text2 and so on. These become part of the: Feature=text1,text2... attribute to the NodeName=&lt;node name&gt; entry line in slurm.conf, as indicated in man(5) slurm.conf. The strings also become added attributes to the GresTypes entry of that file. Default: blank.</td>
</tr>
<tr>
<td>GPU devices</td>
<td>File=&lt;device&gt; entries in the file gres.conf</td>
<td>This is only implemented if the number of GPUs has been set by the administrator using Bright Cluster Manager, by setting the basic Slurm client option GPUs. The GPUs option is seen, for example, in Bright View, via the clickpath Grouping→Node categories[default]→Edit→SettingsRoles→slurmclient→GPUs. Each device detected on the host node can be added as an entry. For example: /dev/nvidia0 /dev/nvidia1 and so on. Alternatively, they can be added as a range in one line, for example: /dev/nvidia[0-1] These become part of the: Name=gpu File=&lt;device&gt; lines in gres.conf, as indicated in man(5) gres.conf. Default: blank</td>
</tr>
<tr>
<td>Slots</td>
<td>CPU</td>
<td>Number of logical processors on the node. Default: 1</td>
</tr>
<tr>
<td>Sockets</td>
<td>Sockets</td>
<td>Processor chips on node. If this is defined, then SocketsPerBoard must not be defined. Default: 1</td>
</tr>
<tr>
<td>Cores per socket</td>
<td>CoresPerSocket</td>
<td>Number of cores per socket. Default: 1</td>
</tr>
<tr>
<td>ThreadsPerCore</td>
<td>ThreadsPerCore</td>
<td>Number of logical threads for a single core. Default: 1</td>
</tr>
</tbody>
</table>
### 7.5 Configuring And Running Individual Workload Managers

<table>
<thead>
<tr>
<th>Bright View Option</th>
<th>Slurm Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Boards</td>
<td>Boards</td>
<td>Number of baseboards in a node. Default: 1</td>
</tr>
<tr>
<td>SocketsPerBoard</td>
<td>SocketsPerBoard</td>
<td>Number of processor chips on baseboard. If this is defined, then Sockets must not be defined. Default: 1</td>
</tr>
<tr>
<td>RealMemory</td>
<td>RealMemory</td>
<td>Size of real memory on the node, MB. Default: 1</td>
</tr>
<tr>
<td>NodeHostname</td>
<td>NodeHostname</td>
<td>Default: as defined by Slurm’s NodeName parameter.</td>
</tr>
<tr>
<td>NodeAddr</td>
<td>NodeAddr</td>
<td>Default: as set by Slurm’s NodeHostname parameter.</td>
</tr>
<tr>
<td>State</td>
<td>State</td>
<td>State of the node with user jobs. Possible Slurm values are: DOWN, DRAIN, FAIL, FAILING, and UNKNOWN. Default: UNKNOWN</td>
</tr>
<tr>
<td>Weight</td>
<td>Weight</td>
<td>The priority of the node for scheduling. Default: 1</td>
</tr>
<tr>
<td>Port</td>
<td>Port</td>
<td>Port that slurmd listens to on the compute node. Default: as defined by SlurmdPort parameter. If SlurmdPort is not specified during build: Default: 6818</td>
</tr>
<tr>
<td>TmpDisk</td>
<td>TmpDisk</td>
<td>Total size of Slurm’s temporary filesystem, TmpFS, typically /tmp, in MB. TmpFS is the storage location available to user jobs for temporary storage. Default: 0</td>
</tr>
<tr>
<td>Options</td>
<td>extra options</td>
<td>Extra options that are added to slurm.conf</td>
</tr>
</tbody>
</table>

Further Slurm documentation is available:

- via man pages under /cm/shared/apps/slurm/current/man.
- as HTML documentation in the directory /usr/share/doc/slurm-18.08.8/html
- at the Slurm website at http://slurm.schedmd.com/documentation.html

Slurm is set up with reasonable defaults, but administrators familiar with Slurm can reconfigure the configuration file /cm/shared/apps/slurm/var/etc/slurm.conf using the JavaScript-based configuration generator in /usr/share/doc/slurm-18.08.8/html/configurator.html in a web browser. If the file becomes mangled beyond repair, the original default can be regenerated once again by re-installing the Slurm package, then running the script /cm/shared/apps/slurm/var/cm/cm-restore-db-password, and then running wlm-setup. Care must be taken to avoid duplicate parameters being set in the configuration file—slurmd may not function correctly in such a configuration.

© Bright Computing, Inc.
Running Slurm
Slurm can be disabled and enabled with the wlm-setup tool (section 7.3) during package installation itself. Alternatively, role assignment and role removal also enables and disables Slurm from Bright View (section 7.4.1) or cmsh (section 7.4.2).

The Slurm workload manager runs these daemons:

1. as servers:
   (a) slurmdbd: The database that tracks job accounting. It is part of the slurmdbd service.
   (b) slurmctld: The controller daemon. Monitors Slurm processes, accepts jobs, and assigns resources. It is part of the slurm service.
   (c) munged: The authentication (client-and-server) daemon. It is part of the munge service.

2. as clients:
   (a) slurmd: The compute node daemon that monitors and handles tasks allocated by slurmctld to the node. It is part of the slurm service.
   (b) slurmstepd: A temporary process spawned by the slurmd compute node daemon to handle Slurm job steps. It is not initiated directly by users or administrators.
   (c) munged: The authentication (client-and-server) daemon. It is part of the munge service.

Logs for the daemons are saved on the node that they run on. Accordingly, the locations are:

- /var/log/slurmdbd
- /var/log/slurmd
- /var/log/slurmctld
- /var/log/munge/munged.log

7.5.2 Configuring And Running SGE
There are several flavors of SGE available since around 2010.

The reference implementation for Bright Cluster Manager is Open Grid Scheduler (OGS), and is provided by the Bright Computing repository. References to SGE in the manuals therefore imply OGS. If other flavors of SGE are meant, then it is stated explicitly. For example, in the section for Univa Grid Engine (UGE, section 7.5.3), UGE means UGE, and not SGE.

Configuring SGE
After installation and initialization, SGE has reasonable defaults, with $SGE_ROOT set to /cm/shared/apps/sge/current.

Administrators familiar with SGE can reconfigure it using the template files in /cm/shared/apps/sge/var/cm/templates/, which define the queues, host-groups and parallel environments that the wlm-setup utility uses. To configure the head node for use with SGE, the install_qmaster wrapper script under $SGE_ROOT is run. To configure a software image for use with SGE the install_execd wrapper script under $SGE_ROOT is run.

By default, the SGE application is installed in /cm/shared/apps/sge/current, the SGE documentation in /cm/shared/docs/sge/current, and job examples in /cm/shared/examples/workload/sge/jobscripts/.

The configuration of submission and execution hosts is similar to that of UGE configuration.
Running SGE
SGE can be disabled and enabled with the `wlm-setup` tool (section 7.3) during package installation itself. Alternatively, role assignment and role removal also enables and disables SGE from Bright View (sections 7.4.1) or `cmsh` (section 7.4.2).

The SGE workload manager runs the following two daemons:

1. an `sge_qmaster` daemon running on the head node. This handles queue submissions and schedules them according to criteria set by the administrator.
2. an `sge_execd` execution daemon running on each compute node. This accepts, manages, and returns the results of the jobs on the compute nodes.

Messages from the qmaster daemon are logged under:

```
/cm/shared/apps/sge/current/default/spool/
```

On the associated compute nodes the execution log messages exists (alongside other job tracking files and directories) at:

```
/cm/local/apps/sge/var/spool/node<number>/messages
```

where `node<number>` is the node name, for example:

```
node001, node002 ...
```

SGE Submission And Execution Hosts
In SGE terminology, a `submission host`, or `submit host`, is a node that is allowed to submit jobs. An execution host is a node on which the computational part of the job is executed.

Standard Bright Cluster Manager Use
In a standard Bright Cluster Manager configuration, a `submission host` or `execution host` in SGE are configured as follows:

**Submission host:** If one of the following 2 conditions is met, in node or category properties for the host:

1. Both the `sgeserver` and `login` roles are assigned to the host
   or
2. Both the `sgeclient` and `login` roles are assigned to the host

then the host is managed by Bright Cluster Manager automatically as a submit host in SGE.

**Execution host:** A submit host is typically not configured by administrators to run jobs. Thus, in Bright Cluster Manager, a submit host cannot normally be assigned a `sgeclient` role, in order to run as an execution host. For unusual situations, the advanced configuration directive `AdditionalExecHosts` described next, can be used to allow the submit host to run as an execution host.

Non-standard Bright Cluster Manager Use
For non-standard configurations, where the SGE service or login service is managed outside of Bright Cluster Manager control, advanced configuration directives allow the cluster to be aware of the situation.
1. If the sgeclient role cannot be added, for instance due to the SGE service being managed outside of Bright Cluster Manager, then the submit hosts that are to be execution hosts can be specified in the advanced configuration directive, AdditionalExecHosts, in the cmd.conf file, as follows:

**Example**

```
AdvancedConfig = {"AdditionalExecHosts=node002,login01"}
```

2. If the host has a sgeclient role, but the cluster is configured so that no login role can be added to a node that is be a submit host, for instance due to the login service being managed outside of Bright Cluster Manager, then the execution hosts that are to be submit hosts can be specified in the advanced configuration directive, AdditionalSubmitHosts, in the cmd.conf file, as follows:

**Example**

```
AdvancedConfig = {"AdditionalSubmitHosts=node002,login01"}
```

### 7.5.3 Installing, Configuring, And Running UGE

The workload manager, Univa Grid Engine (UGE), is a flavor of SGE (section 7.5.2) developed by Univa. Bright Cluster Manager 8.2 supports integration with UGE versions 8.2.0 and higher at the time of writing of this section (October 2017). Older versions can also be supported, as detailed in the Bright Computing Knowledge Base (https://kb.brightcomputing.com).

UGE should be picked up directly from the Univa website at http://www.univa.com.

The installation and integration of UGE version 8.2.0 into Bright Cluster Manager 8.2 can be carried out as in the following steps:

1. These UGE tar.gz file collections should be downloaded from the Univa website to a directory on the head node:
   - Binary files:
     - The 64-bit bundle
       ```
ge-<uge_ver>-bin-lx-amd64.tar.gz
```
     - or the 32-bit bundle
       ```
ge-<uge_ver>-bin-lx-x86.tar.gz
```
   - Common files:
     ```
ge-<uge_ver>-common.tar.gz
```

   Here `<uge_ver>` is the UGE version, for example: 8.2.0

   To avoid installation issues, a check should be done to ensure that during the download, the tar.gz files have not been renamed, or that their names not been changed to upper case. Both packages must be located in the same directory before installation.

   If a failover setup already exists, then the installation should be done on the active head node in the steps that follow.

2. The cm-uge package should be installed from the Bright Computing repository:

   **Example**

   - For RHEL-based distributions:
     ```
     [root@bright82 ~]# yum install cm-uge
     ```

   - For SLES distributions:

© Bright Computing, Inc.
The package installs, amongst others, the following template files under /cm/shared/apps/uge/var/cm/:

(a) An environment module template, uge.module.template
(b) An installation configuration template, inst_template.conf.template
(c) Some other template files for a default GE configuration, under the directory templates

The templates decide the configuration of UGE (section “Configuring UGE”, page 260). The original files 2a and 2b, with the .template suffix, should never be modified by the administrator. The administrator can change the parameters affected by the installation configuration from their default values by copying these original files to the same directory without the .template suffix, and then editing the .conf file:

Example

[root@bright82 ~]# cd /cm/shared/apps/uge/var/cm
[root@bright82 cm]# cp inst_template.conf.template inst_template.conf
[root@bright82 cm]# vi inst_template.conf

The copied file, inst_template.conf, can be changed by the administrator to decide the location of UGE software and how UGE handles jobs, if needed. The changed file overrides the settings suggested by the .template file, when the Bright Cluster Manager utility wlm-setup runs in the next step of the installation procedure.

Some of the values in the key-value pairs in the file are enclosed by percentage signs, %. Such flagged values should not be modified, since they are replaced by Bright Cluster Manager values by wlm-setup during installation.

Values not enclosed by percentage signs are not replaced by wlm-setup. Such unflagged values can, if needed, be tuned in the copied file by the administrator. These values are then kept by wlm-setup during installation, and used when UGE is run.

3. wlm-setup is run. The directory where the downloaded UGE files are located is specified with the -a option.

Example

[root@bright82 ~]# wlm-setup -w uge -s -a /root/uge

The wlm-setup step is actually a wrapper here. It runs the inst_sge script that comes with UGE, as one of its actions. The inst_sge script should not normally be run directly in a cluster that runs Bright Cluster Manager.

4. The nodes are rebooted. The UGE command qhost then displays an output similar to:

Example

[root@bright82 ~]# module load uge
[root@bright82 ~]# qhost
HOSTNAME  ARCH  NCPUS  NSOC  NCOR  NTHR  NLOAD
-------------------------------------------------

© Bright Computing, Inc.
global  -  -  -  -  -  -  ...
node001 lx-amd64  8  1  1  8  0.01 ...
node002 lx-amd64  8  1  1  8  0.01 ...
node003 lx-amd64  8  1  1  8  0.01 ...
node004 lx-amd64  8  1  1  8  0.01 ...

The output in the preceding example has been truncated for this manual, for convenience.

**Configuring UGE**

After installation and initialization, UGE has reasonable defaults, with \$SGE_ROOT set to `/cm/shared/apps/uge/current`.

By default, the UGE application is installed in `/cm/shared/apps/uge/current`, and job examples are kept in `/cm/shared/examples/workload/sge/jobscripts/`.

**Running UGE**

The UGE workload manager runs the following two daemons:

1. an `sge_qmaster` daemon running on the head node. This handles queue submissions and schedules them according to criteria set by the administrator.

2. an `sge_execd` execution daemon running on each compute node. This accepts, manages, and returns the results of the jobs on the compute nodes.

Messages from the qmaster daemon are logged under:

`/cm/shared/apps/uge/current/default/spool/`

On the associated compute nodes the execution log messages exists, alongside other job tracking files and directories, at:

`/cm/local/apps/uge/var/spool/node<number>/messages`

where `node<number>` is the node name, for example:

node001, node002 ...

**UGE Submission, Administrative And Execution Hosts**

UGE is still similar to SGE in many ways, but is evolving.

In UGE terminology:

- A *submission host*, or *submit host*, is a node that is allowed to submit jobs.

- An *execution host*, or *exec host*, is a node on which the computational part of the job is executed.

- An *administrative host*, or *admin host*, is a node that can carry out administrative Grid Engine commands.

**Standard Bright Cluster Manager Use**

In a standard Bright Cluster Manager configuration, a *submission*, *administrative* or *execution* host in UGE is configured with the following `ugeclient` role properties:

1. Admin Host – adds the node to a list of administrative hosts in UGE,

2. Exec Host – adds the node to a list of execution hosts in UGE,
3. Submit Host – adds the node to a list of submission hosts in UGE.

The nodes that are assigned a ugeserver role are always automatically added to the administrative and submission host lists.

Non-standard Bright Cluster Manager Use

For non-standard configurations, where the UGE service or login service is managed outside of Bright Cluster Manager control, advanced configuration directives allow the cluster to be aware of the situation.

1. If the ugeclient role cannot be added, for instance due to the UGE service being managed outside of Bright Cluster Manager, then the additional submit hosts can be specified in the advanced configuration directive, AdditionalExecHosts, in the cmd.conf file:

   Example

   AdvancedConfig = {"AdditionalExecHosts=node002,login01"}

2. If the ugeclient role cannot be added, for instance due to the login service being managed outside of Bright Cluster Manager, then the execution hosts can be specified in the advanced configuration directive, AdditionalSubmitHosts, in the cmd.conf file:

   Example

   AdvancedConfig = {"AdditionalSubmitHosts=node002,login01"}

Parallel Environments

A parallel environment (PE) allows the configuration of applications that use shared or distributed memory. The main—but not sole—purpose of a PE is to provide a job environment within which MPI applications can run, using various MPI libraries. PEs can be provided by the cluster software or hardware vendor, as well as distributed with UGE itself. Default PEs are provided for various MPI libraries, such as Open MPI, MVAPICH, and so on. The default PEs are automatically added to UGE when wlm-setup installs UGE.

The CMDaemon front ends, i.e. cmsh or Bright View, can be used to add, remove, or modify the existing PE. Each node or node category with a ugeserver role has a Parallel Environments submode. In the Parallel Environments submode, each PE is represented as a separate object with associated properties. For example, the following session displays Open MPI properties (some text elided):

Example

[root@bright82 ~]# cmsh
[bright82]% device roles bright82
[bright82->device[bright82]->roles]% assign ugeserver; commit
[bright82->...->roles[ugeserver]]% parallelenvironments
[bright82->...->roles[ugeserver]->parallelenvironments]% show openmpi

Parameter Value
--------------------------- ---------------------------
Accounting Summary yes
Allocation Rule $round_robin
Control Slaves yes
Daemon Forks Slaves yes
Extra Parameter
Job Is First Task no
Master Forks Slaves no

© Bright Computing, Inc.
262 Workload Management

Name: openmpi
Slots: 999999
Start Procedure Arguments: NONE
Stop Procedure Arguments: NONE
Urgency Slots: min
User Lists: NONE
User Lists: NONE

The values shown in the preceding example are defaults.

Database Recovery

By default, Bright Cluster Manager configures UGE to use flat file, or classic spooling as its format for the sge_qmaster spool. Typically this can handle thousands of jobs a day without running into performance issues, and it is the current (May 2015) recommendation for stability reasons.

For larger clusters it is possible to use Berkeley DB as the spooling format. This comes with a number of utilities, some of which can be useful for UGE spool maintenance and debugging, such as db_recover, db_verify, and others.

The Berkeley DB utilities that come with UGE are modified versions of the ones that are shipped with the parent distribution Berkeley DB packages. The UGE versions include extra sanity tests for the UGE spool database, and it is best to use them instead of the standard versions. The utilities can be found in on the head node under:

/cm/shared/apps/uge/current/utilbin/lx-amd64/

This directory is not added to the $PATH when loading the UGE environment with module load uge.

The full Berkeley DB documentation for these tools is part of the UGE distribution. A local HTML format copy of the documentation is available at:

/cm/shared/apps/uge/current/doc/berkeleydb/db_*.html

GPU And MIC Management

When managing host resources such as GPU or Xeon Phi (MIC) PCI-e cards\(^1\), UGE 8.1.0 and higher allows the accelerators to be configured via a resource map, RSMAP. This is a complex (man (5) complex) value type defined on the host. RSMAP values restrict how much of a resource is used concurrently from a host, attaches identifiers to the resource used, and assigns the identifiers to the jobs when they get dispatched to that host. Bright Cluster Manager automatically configures UGE execution hosts with gpu and phi RSMAP resources.

The ugeclient role can configure GPUs by using the following cmsh parameters:

- GPU devices: a list of GPU names that are attached to a job by UGE.
- Gpus: the number of GPUs on a host.

The GPU devices parameter has a higher priority than Gpus, so that if names are set in GPU devices, then they are always used.

GPUs can also be bound to particular CPU sockets and CPU cores, using a topology mask.

The mask is defined with a set of characters. In principle, the mask is applied to the computing units available for a device. The computing units correspond to processing units, and are grouped as CPU

\(^1\)Integration of PCI-e MIC cards is discontinued from Bright Cluster Manager 8.2 onwards, but UGE support for the cards via RSMAP attributes continues. This means that the administrator typically would download the MIC MPSS packages, compile the driver, install the images manually, and after getting the MICs to work, would integrate the MIC with the appropriate RSMAP attributes.

© Bright Computing, Inc.
sockets, cores, and hardware threads. These correspond to the letters S, C, T. In the mask. An uppercase character allows a device to use a computing unit, while a lowercase character forbids it, as indicated by the following table:

<table>
<thead>
<tr>
<th>Unit To Mask</th>
<th>Enable Unit with</th>
<th>Disable Unit with</th>
</tr>
</thead>
<tbody>
<tr>
<td>Socket S</td>
<td>s</td>
<td>s</td>
</tr>
<tr>
<td>Core C</td>
<td>c</td>
<td>c</td>
</tr>
<tr>
<td>Hardware Thread T</td>
<td>t</td>
<td>t</td>
</tr>
</tbody>
</table>

Table 7.5.3: Mask Composition

In practice, s and t are currently (May 2015) ignored, so that only cores can be masked. Some mask examples:

- **SccSCC**: This is a two-core two-socket system. The first socket cannot be used, while both cores on the second socket can be used.
- **SCcCCScCC**: This is a four-core two-socket system. The first socket has its first, third, and fourth cores available, and the second socket has its third and fourth cores available.

Some configuration examples from cmsh and the corresponding RSMAP complex attribute values on the host:

- **Naming 2 GPUs**:

```
[root@bright82 ~]# cmsh
[bright82] # category roles default
[bright82->device[bright82]->roles]# assign ugeclient; commit
[bright82->device[bright82]->roles[ugeclient]]# show | grep -i gpu
GPU devices
gpu0
gpu1
GPUs
2
```

The RSMAP attribute value is then:

```
GPU=2(gpu0 gpu1)
```

- **Allow GPUs to use only different sockets**:

```
[bright82->device[bright82]->roles[ugeclient]]# show | grep -i gpu
GPU devices
gpu0:SCCCScccc
gpu1:ScccSCCCC
GPUs
0
```

The RSMAP attribute value is then:

```
GPU=2(gpu0:SCCCScccc
gpu1:ScccSCCCC)
```

- **Allow Xeon Phi resources to set up sockets in a spaced-out way**:

```
[bright82->device[bright82]->roles[ugeclient]]# show | grep mic
MIC devices
mic0:SCccc
mic1:SccCc
mic2:SccCc
mic3:SccCc
```

As is seen here, MICs are set up very much like GPUs, and also within the ugeclient role. The only differences are that the parameters that are set differ according to the accelerator used:

- while for GPUs topology masks are set for the `GPU devices` parameter, for MICs topology masks are set for the `MIC devices` parameter

© Bright Computing, Inc.
while for GPUs the number of GPUs is manually specified in the gpus parameter, for MICs the number of MICs is counted automatically and there is no equivalent parameter.

The RSMAP attribute value in this example is then:

MIC=4(mic0:SCccc mic1:ScCcc mic2:SccCc mic3:ScCcC)

Further detailed information about RSMAP and topology mask usage can be found in the *Univa Grid Engine Administrator’s Guide*.

### 7.5.4 Configuring And Running PBS Pro

#### PBS Pro Versions

Bright Cluster Manager provides two versions of PBS Pro:

- PBS Pro. This is the commercial version.
- PBS Pro Community Edition (or CE). This is the edition with community support (http://pbspro.org).

Both versions can be installed as a selection option during Bright Cluster Manager 8.2 installation, at the point when a workload manager must be selected (figure 3.26 of the *Installation Manual*). Alternatively they can be installed later on, when the cluster has already been set up.

When no particular edition is specified in the Bright Cluster Manager documentation, then the text is valid for both editions.

If Bright Cluster Manager has already been set up without PBS Pro, then the wlm-setup tool (section 7.3) should be used to install and initialize a PBS Pro edition.

#### PBS Pro Versions

Bright Cluster Manager is integrated with multiple versions of the PBS Pro Commercial and PBS Pro CE workload managers. The package names for these versions follow the pattern:

- For PBS Pro Commercial:
  - `pbspro<upstream major version number>`
  - `pbspro<upstream major version number>-client`

- for PBS Pro CE:
  - `pbspro<upstream major version number>-ce`
  - `pbspro<upstream major version number>-ce-client`

At the time of writing of this section (August 2019) `<major version number>` has the possible values of 18 or 19. The most recent packages available for the variants at the time of writing are therefore:

- For the Commercial variant: `pbspro19` and `pbspro19-client`
- For the CE variant: `pbspro19-ce` and `pbspro19-ce-client`

Bright Cluster Manager provides a similar level of integration for all of these packages (Commercial, CE, version 18, version 19). It is up to the cluster administrator to decide which version is set up.

Installing a PBS Pro package without specifying a version number, for example using the package name `pbspro` or `pbspro-ce`, installs the default version number. For Bright Cluster Manager version 8.2 the corresponding PBS Pro package version number default is 18.

The installed package can be removed with a package manager such as YUM without specifying a version number.
To switch to a different (typically more recent) package, the administrator has to uninstall any existing package first. This is to avoid a conflict between the versions and variants.

For example, an existing pbspro-ce package can be removed and a new pbspro-ce package can be installed as follows (assuming the administrator is using YUM, and that the PBS Pro workload manager is already disabled):

**Example**

```
[root@bright82 ~]# yum remove pbspro-ce pbspro-ce-client
[root@bright82 ~]# yum remove --installroot=/cm/images/default-image pbspro-ce-client
[root@bright82 ~]# yum install pbspro19-ce pbspro19-ce-client
[root@bright82 ~]# yum install --installroot=/cm/images/default-image pbspro19-ce-client
```

Similarly, to update existing pbspro packages (again, assuming that the administrator is using YUM, and that the PBS Pro workload manager is already disabled):

**Example**

```
[root@bright82 ~]# yum remove pbspro pbspro-client
[root@bright82 ~]# yum remove --installroot=/cm/images/default-image pbspro-client
[root@bright82 ~]# yum install pbspro19 pbspro19-client
[root@bright82 ~]# yum install --installroot=/cm/images/default-image pbspro19-client
```

The pbspro and pbspro-ce packages that Bright Computing provides should be used instead of any other available versions.

**Installing PBS Pro**

After package installation via the package manager, as described in the preceding section, both PBS Pro commercial and PBS Pro CE can be installed and initialized to work with Bright Cluster Manager via `wlm-setup`:

**Example**

```
[root@bright82 ~]# wlm-setup -w pbspro -s
```

or

```
[root@bright82 ~]# wlm-setup -w pbspro-ce -s
```

Installing one automatically disables the other.

The software components are installed and initialized by default under the `PBS_HOME` directory, which is either:

- `/cm/shared/apps/pbspro/current`
- `/cm/shared/apps/pbspro-ce/current`

as appropriate.

Users must load the pbspro environment module, which sets `$PBS_HOME` and other environment variables, in order to use either edition of PBS Pro.

**Upgrading PBS Pro**

Some users may already have pbspro or pbspro-ce deployed in Bright Cluster Manager version 8.2, and running PBS Pro version 18. If they would like to upgrade to a more recent version (pbspro19 or pbspro19-ce), then the following procedure can be followed for pbspro-ce:

**Example**
266 Workload Management

[root@bright82 ~]# wlm-setup -d -w pbspro-ce
[root@bright82 ~]# yum remove pbspro-ce pbspro-ce-client
[root@bright82 ~]# yum remove --installroot=/cm/images/default-image pbspro-ce-client
[root@bright82 ~]# yum install pbspro19-ce pbspro19-ce-client
[root@bright82 ~]# yum install --installroot=/cm/images/default-image pbspro19-ce-client
[root@bright82 ~]# wlm-setup -e -w pbspro-ce

Similarly, for pbspro:

Example

[root@bright82 ~]# wlm-setup -d -w pbspro
[root@bright82 ~]# yum remove pbspro pbspro-client
[root@bright82 ~]# yum remove --installroot=/cm/images/default-image pbspro-client
[root@bright82 ~]# yum install pbspro19 pbspro19-client
[root@bright82 ~]# yum install --installroot=/cm/images/default-image pbspro19-client
[root@bright82 ~]# wlm-setup -e -w pbspro

An upgrade replaces custom configurations, so that configuration settings such as queues are lost.

PBS Pro Configuration


By default, PBS Pro examples are available under the directory /cm/shared/examples/workload/pbspro/jobscripts/

Some PBS Pro configuration under Bright Cluster Manager can be done using roles. The roles are the same for both the commercial and the CE editions:

- In Bright View the roles setting allows the configuration of PBS Pro client and server roles.
  - For the PBS Pro server role, the role is enabled, for example on a head node bright82, via a clickpath of: Devices → Head Nodes → bright82 → Settings → Roles → Add → PBS pro server role
    * Within the role window for the server role, its installation path and server spool path can be specified.
  - For the PBS Pro client role, the role is enabled along a similar clickpath, just ending at PBS pro client role. The number of slots, GPUs and other properties can be specified, and queues can be selected.

- In cmsh the client and server roles can be managed for the individual nodes in device mode, or managed for a node category in category mode.

Example

[root@bright82 ~]# cmsh
[bright82]% device roles bright82
[bright82->device[bright82]->roles]% assign pbsproserver; commit
[bright82->device[bright82]->roles[pbsproserver]]% show

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>External Server</td>
<td>no</td>
</tr>
<tr>
<td>Name</td>
<td>pbsproserver</td>
</tr>
<tr>
<td>Provisioning associations</td>
<td>&lt;0 internally used&gt;</td>
</tr>
<tr>
<td>Revision</td>
<td></td>
</tr>
<tr>
<td>Type</td>
<td>PbsProServerRole</td>
</tr>
<tr>
<td>Prefix</td>
<td>/cm/shared/apps/pbspro/current</td>
</tr>
</tbody>
</table>

© Bright Computing, Inc.
Cloudbursting with cluster extension may need special handling for PBS Pro. If this feature is needed, then the administrator should contact Bright Computing support via the website https://support.brightcomputing.com.

Further configuration of PBS Pro is done using its qmgr command and is covered in the PBS Pro documentation.

**Running PBS Pro**

PBS Pro runs the following four daemons:

1. A pbs_server daemon running, typically on the head node. This handles submissions acceptance, and talks to the execution daemons on the compute nodes when sending and receiving jobs. It writes logs to the var/spool/server_logs/ directory that is under /cm/shared/apps/pbspro or /cm/shared/apps/pbspro-ce on the nodes it is associated with. Queues for this service are configured with the qmgr command.

2. A pbs_sched scheduler daemon, also typically running on the head node. It writes logs to the var/spool/sched_logs/ directory under /cm/shared/apps/pbspro or /cm/shared/apps/pbspro-ce.

3. A pbs_mom execution daemon running on each compute node. This accepts, manages, and returns the results of jobs on the compute nodes. It writes logs on the head node to the var/spool/mom_logs/ directory under /cm/shared/apps/pbspro or /cm/shared/apps/pbspro-ce, and to /cm/local/apps/pbspro/var/spool/mom_logs/ on the compute nodes.

4. A pbs_comm communication daemon usually running on the head node. This handles communication between PBS Pro daemons, except for server-to-scheduler and server-to-server daemons communications. It writes logs to the head node under the relative directory var/spool/comm_logs/, under /cm/shared/apps/pbspro or /cm/shared/apps/pbspro-ce, and writes logs to the compute nodes under /cm/local/apps/pbspro/var/spool/comm_logs/.

**Running PBS Pro On Cluster Extension**

When PBS Pro is set up with wlm-setup or during the installation of the head node, then the pbsproclient role is assigned by default to the default node category only. In order to add cloud nodes to PBS Pro, the administrator can assign the pbsproclient role manually.

There are two types of PBS Pro configuration in this case. The configurations can be applied to both the commercial and to the community editions.

1. pbs_mom daemons on cloud compute nodes communicate to the pbs_server directly.

   This scenario is suited to a non-VPN setup where cloud nodes have addresses on the same IP subnet for the cloud and for the on-premise parts of the cluster. Usually this kind of setup is used with Amazon DirectConnection or Azure ExpressRoute. In order to add new cloud nodes, the administrator just needs to assign the pbsproclient role to the cloud node category or the cloud nodes directly.

2. pbs_mom daemons on cloud compute nodes communicate to the pbs_server via a separate pbs_comm server. Bright Computing recommends the use of the cloud-director for this purpose.

   This can be useful if cluster extension is configured with a VPN tunnel setup. In this case the pbs_mom running on a cloud node communicates with the pbs_server by using the VPN connection, and the communication traffic goes via an OpenVPN server. The OpenVPN connection adds overhead on the cloud-director where the OpenVPN daemon runs. If the traffic is routed via pbs_comm running on cloud-director, then the OpenVPN server is not used. This is because
pbs_comm daemon on the cloud director resolves the cloud pbs_mom addresses with cloud IP addresses, while pbs_server resolves pbs_comm on the cloud director by using the VPN tunnel IP.

In order to configure PBS Pro to pass the communication traffic via pbs_comm, the administrator should assign pbsproclient roles to not only the compute cloud nodes, but also to the cloud-director. On the cloud director, the administrator should enable pbs_comm daemon to start, and pbs_mom daemon to not start, automatically. These actions are done in the commsettings and momsettings submodes of pbsproclient role.

Further configuration that should be carried out is to set the commrouters parameter on the cloud director to master, and set the leafrouters parameter on the compute cloud nodes to the hostname of the cloud director.

For example (some text elided):

Example

```
[root@bright82 ~]# cmsh
[bright82]% category roles cloud-nodes
[bright82->category[cloud-nodes]->roles]% assign pbsproclient
[bright82->category[cloud-nodes]->roles]% set queus cloudq
[bright82->...]->roles[pbsproclient]->momsettings]% momsettings
[bright82->...]->roles[pbsproclient]->momsettings]% set leafrouters director
[bright82->...]->roles[pbsproclient]->momsettings]% commit
[bright82->...]->roles[pbsproclient]->momsettings]% device roles director
[bright82->...]->roles[pbsproclient]% assign pbsproclient
[bright82->...]->roles[pbsproclient]% momsettings
[bright82->...]->roles[pbsproclient]->momsettings]% set startmom no
[bright82->...]->roles[pbsproclient]->momsettings]% ..
[bright82->...]->roles[pbsproclient]->momsettings]% commsettings
[bright82->...]->roles[pbsproclient]->commsettings]% set startcomm yes
[bright82->...]->roles[pbsproclient]->commsettings]% set commrouters master
[bright82->...]->roles[pbsproclient]->commsettings]% commit
[bright82->...]->roles[pbsproclient]->commsettings]%
```

7.5.5 Installing, Configuring, And Running LSF

IBM prefers to make LSF available directly from their Passport Advantage Online website, which is why it is not available as an option in the installation carried out by Bright Cluster Manager 8.2 in figure 3.26 of the Installation Manual.

Installing LSF

The workload manager LSF versions 9 or 10 are installed and integrated into Bright Cluster Manager 8.2 with the following steps:

1. The following LSF files should be downloaded from the IBM web site into a directory on the head node:
   - Installation package: lsf<lsf_ver>_lsfinstall_linux_<cpu_arch>.tar.Z
   - Distribution package: lsf<lsf_ver>_linux<kern_ver>-glibc<glibc_ver>_<cpu_arch>.tar.Z
   - Documentation package: lsf<lsf_ver>_documentation.tar.Z
   - License file: platform_lsf_<lic_type>_entitlement.dat

Here:

   - <lsf_ver> is the LSF version, for example: 9.1.1

© Bright Computing, Inc.
7.5 Configuring And Running Individual Workload Managers

- `<kern_ver>` is the Linux kernel version, for example: 2.6
- `<glibc_ver>` is the glibc library version, for example: 2.3
- `<cpu_arch>` is the CPU architecture, for example: x86_64
- `<lic_type>` is the license type, for example: std

A check should be done to ensure that the tar.Z files have not been renamed or had their names changed to lower case during the download, in order to avoid installation issues. All 4 files must be in the same directory before installation.

In case of an existing failover setup, the installation is done on the active head node.

2. The cm-lsf package must be installed from the Bright Computing repository:

```
[root@bright82 ~]# yum install cm-lsf #RHEL-based distributions
```

or

```
[root@bright82 ~]# zypper install cm-lsf #SLES distributions.
```

The cm-lsf package contains a template environment module file and an installation configuration file. The installation configuration file may be tuned by administrator if required. It is passed to the lsfinstall script distributed with LSF, which is be executed by `wlm-setup` during setup. To change the default values in the installation configuration file, the administrator should copy the template file first and then change the copied file:

```
[root@bright82 ~]# cd /cm/shared/apps/lsf/var/cm/
[root@bright82 cm]# cp install.config.template install.config
[root@bright82 cm]# vi install.config
```

The install.config file can be changed by the administrator to decide the location of LSF software and some other LSF configuration options, if needed. The changed file overrides the settings suggested by the .template file, when the Bright Cluster Manager utility `wlm-setup` runs in the next step of the installation procedure.

A few of the values in the key-value pairs in the file are enclosed by percentage signs, %. Such “flagged” values should not be modified, since they are replaced by Bright Cluster Manager values by `wlm-setup` during installation.

Values not enclosed by percentage signs are left alone by `wlm-setup`. Such “unflagged” values can, if needed, be tuned in the file copy by the administrator. These values are then kept by `wlm-setup` during installation, and used when LSF is run.

3. `wlm-setup` is run. The directory where the LSF files were downloaded is specified with the `-a` option.

**Example**

```
[root@bright82 ~]# wlm-setup -w lsf -s -a /root/lsf
```

4. The nodes are then rebooted, and the LSF command `bhosts` then displays an output similar to:

**Example**
### Workload Management

[Shell Command]

```bash
[root@bright82 ~]# module load lsf
[root@bright82 ~]# bhosts

<table>
<thead>
<tr>
<th>HOST_NAME</th>
<th>STATUS</th>
<th>JL/U</th>
<th>MAX</th>
<th>NJOBS</th>
<th>RUN</th>
<th>SSUSP</th>
</tr>
</thead>
<tbody>
<tr>
<td>bright82</td>
<td>ok</td>
<td>-</td>
<td>2</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>head2</td>
<td>ok</td>
<td>-</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>node001</td>
<td>ok</td>
<td>-</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>node002</td>
<td>ok</td>
<td>-</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>node003</td>
<td>unavail</td>
<td>-</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>node004</td>
<td>closed</td>
<td>-</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>node005</td>
<td>ok</td>
<td>-</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
```

The output in the preceding example has been truncated for this manual, for convenience.

The installation status can be checked with:

```bash
[root@bright82 ~]# /etc/init.d/lsf status
Show status of the LSF subsystem
lim (pid 21138) is running...
res (pid 17381) is running...
sbatchd (pid 17383) is running...
```

while default queues can be seen by running:

```bash
[root@bright82 ~]# module load lsf
[root@bright82 ~]# bqueues

<table>
<thead>
<tr>
<th>QUEUE_NAME</th>
<th>PRIO</th>
<th>STATUS</th>
<th>MAX</th>
<th>JL/U</th>
<th>JL/P</th>
</tr>
</thead>
<tbody>
<tr>
<td>owners</td>
<td>43</td>
<td>Open:Active</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>priority</td>
<td>43</td>
<td>Open:Active</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>night</td>
<td>40</td>
<td>Open:Active</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>chkpt_rerun_qu</td>
<td>40</td>
<td>Open:Active</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>short</td>
<td>35</td>
<td>Open:Active</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>license</td>
<td>33</td>
<td>Open:Active</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>normal</td>
<td>30</td>
<td>Open:Active</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>interactive</td>
<td>30</td>
<td>Open:Active</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>idle</td>
<td>20</td>
<td>Open:Active</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>
```

The output in the preceding example has been truncated for this manual, for convenience.

### Configuring LSF

**LSF server configuration:** After LSF is set up with `wlm-setup`, the following CMDaemon settings can be modified for the LSF server role:

- **Prefix:** this sets the path to the root location of the LSF installation. The default value is `/cm/shared/apps/lsf/current`.
- **Var:** this sets the path to the var directory of LSF. The default value is `/cm/shared/apps/lsf/var`.
- **External Server:** a value of yes means that LSF server daemons are running on an external server that is not managed by Bright Cluster Manager.
- **Cgroups:** this is a submode that contains LSF-related cgroups settings.

The cgroups settings provide the following parameters that affect LSF behaviour:

- **Automount:** if yes, then the workload manager tries to mount a subsystem if it is not mounted yet. The default value is No.
• **Job Cgroup Template:** this is the template relative job cgroup path. The token \$CLUSTER specified in this template path is replaced with the actual LSF cluster name, and \$JOBID is replaced by the job ID. The path is used by the Bright Cluster Manager monitoring system in order collect job metrics from the cgroups. By default, the path is set to "lsf/$CLUSTER/job.$JOBID.*".

• **Process Tracking:** if yes, then processes are tracked based on job control functions such as: termination, suspension, resume, and other signaling. These are used on Linux systems that support the freezer subsystem under cgroups. The parameter sets LSF_PROCESS_TRACKING in lsf.conf.

• **Linux Cgroup Accounting:** if yes, then LSF tracks processes based on CPU and memory accounting. This is for Linux systems that support the memory and cpuacct subsystems under cgroups. Once enabled, this parameter takes effect for new jobs. The parameter sets LSF_LINUX_CGROUP_ACCT in lsf.conf.

If this parameter and Process Tracking are both enabled, then they take precedence over the parameters LSF_PIM_LINUX_ENHANCE and EGO_PIM_SWAP_REPORT in lsf.conf.

• **Mount Point:** specifies a path where cgroups is mounted. It only makes sense to set this when the location is not standard for the operating system.

• **Resource Enforce:** If yes, then resource enforcement is carried out through the Linux memory and cpuset subsystems under cgroups. This is for Linux systems with cgroup support. The parameter sets LSB_RESOURCE_ENFORCE in lsf.conf.

These settings can be modified as follows:

• **Within Bright View:** For example for a head node bright82, via a clickpath of Devices → Head Nodes → bright82 → Settings → Roles → LSF server role

• **Within cmsh:** For a particular category in the category mode, or a particular device in the device mode, the roles submode is chosen. Within the roles submode, the lsfserver parameter can be set. The following example shows how to set the LSF prefix parameter for the default category.

**Example**

```
[root@bright82~]# cmsh
[bright82]# category use default; roles
[bright82->category[default]->roles]% use lsfserver
[bright82->...[lsfserver]]% set prefix /cm/shared/apps/lsf/current2
[bright82->...[lsfserver*]]% commit
```

**LSF client configuration:** After installation, the following CMDaemon settings can be specified for the LSF client role:

• **Queues** A restricted list of queues named “qname1”, “qname2” and so on can be set using a command syntax like this instead:

  ```
  set queues <qname1> [<qname2> ...]
  ```

  Alternatively, these, and other queues can be added or deleted using Bright View (section 7.6.2) or cmsh (section 7.7.2).
• **Options**  The options that can be set for a regular node node001, via figure 7.4, reached via a
clickpath of

Devices→Nodes[node001]→Edit→Settings→Roles→Add→LSF client role are:

- **Slots**: The number of CPUs per node. By default LSF tries to determine the value automatically. If the number of slots is set to 0, then the node becomes an LSF submit host, so that no jobs can run on the host, but users can submit their jobs from this host using the LSF utilities.
- **GPUs**: The number of GPUs per node
- **Queues**: All queues.
- **Advanced button**: This brings up a dialogue that allows GPU devices to be added.
- **Node Customizations**: LSF node custom properties (section 7.11).

From cmsh these properties are accessible from within the appropriate node or category roles submode (section 7.4.2).

**Submission hosts**: By default, all compute nodes to which the lsfclient role is assigned, also become submission hosts. This happens because when the Slots parameter is not set to 0, then Bright Cluster Manager configures the node as a server (in the LSF terminology). This means that LSF users can submit their jobs from those compute nodes.

If the administrator wants to allow users to submit their jobs from a non-compute node—for example from a login node—then the following steps can be followed to configure submission:

1. The lsfclient role is assigned to that node and the parameter Slots is set to 0 in the role.
2. The lsf service is stopped by the administrator using cmsh or Bright View. It must not be stopped by system utilities such as systemctl, because Bright Cluster Manager starts the service again automatically if the lsfclient role has been assigned to the node.
3. The LSF cluster <cluster name> has a configuration file /cm/shared/apps/lsf/var/conf/lsf.<cluster name> inside which the administrator must specify a value for model and a value for type for the node. These values are specified in the appropriate columns of the Host section of the configuration file. The possible values that can be specified are defined in the appropriate section of /cm/shared/apps/lsf/var/conf/lsf.shared. Specifying the values is needed because the lsf service is stopped, and LSF cannot then pick up those values from the service.

© Bright Computing, Inc.
4. On the LSF master node, the administrator must run `lsconfig reconfig` in order to apply the change in the cluster configuration file.

Configuring a node according to the preceding steps then allow users to submit their jobs from that node, but none of the submitted jobs gets scheduled to run on that node.

**Data-Aware Scheduling With `cmsub`:** The ability to use `cmsub` to submit jobs to cloud nodes, as described in section 4.7 of the *User Manual*, is called *data-aware scheduling*. Data-aware scheduling is enabled for LSF as follows:

1. In order to run the cloud prolog script as the root user, `/etc/lsf.sudoers` must be configured. By default this file does not exist. Its ownership must be root. It must also have read-write permissions for root only, i.e. its unix permissions can be set with `chmod 600 /etc/lsf.sudoers`. To execute the `PRE_EXEC` script as root, `LSB_PRE_POST_EXEC_USER=root` should be set in `lsf.sudoers`:

   ```bash
   [root@bright82 ~]$ echo "LSB_PRE_POST_EXEC_USER=root" >> /etc/lsf.sudoers
   [root@bright82 ~]$ chmod 600 /etc/lsf.sudoers
   ```

2. The changes are applied by restarting the hosts:

   ```bash
   [root@bright82 ~]$ badmin hrestart all
   ```

**Further configuration:** For further configuration the *Administering Platform LSF* manual provided with the LSF software should be consulted.

**Running LSF**

Role assignment and role removal enables and disables LSF from Bright View (sections 7.4.1) or `cmsh` (section 7.4.2). An active LSF master (typically, but not necessarily on a head node) has the following LSF-related processes or services running on it:

<table>
<thead>
<tr>
<th>Process/Service</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>res</td>
<td>Remote Execution Server*</td>
</tr>
<tr>
<td>sbatchd</td>
<td>client batch job execution daemon*</td>
</tr>
<tr>
<td>mbatchd</td>
<td>master batch job execution daemon</td>
</tr>
<tr>
<td>eauth</td>
<td>External Authentication method</td>
</tr>
<tr>
<td>lim</td>
<td>Load Information Manager*</td>
</tr>
<tr>
<td>pim</td>
<td>Process Information Manager*</td>
</tr>
<tr>
<td>pem</td>
<td>Process Execution Manager*</td>
</tr>
<tr>
<td>vemkd</td>
<td>Platform LSF Kernel Daemon</td>
</tr>
<tr>
<td>egosc</td>
<td>Enterprise Grid Orchestrator service controller</td>
</tr>
<tr>
<td>mbschd</td>
<td>master batch scheduler daemon</td>
</tr>
</tbody>
</table>

*These services/processes run on compute nodes.

Non-active LSF-masters running as compute nodes run the processes marked with an asterisk only. Logs for LSF processes and services are kept under `/cm/shared/apps/lsf/log/` (or `$LSF_TOP/log/` if the value of `$LSF_TOP` during installation is other than the recommended `/cm/shared/apps/lsf/`).
7.6 Using Bright View With Workload Management

Viewing the workload manager services from Bright View is described in section 7.4.3. The HPC (High Performance Computing) icon, which looks like a speedometer, is the Bright View resource that allows access to:

- jobs
- queues

These items are described next.

7.6.1 Jobs Display And Handling In Bright View

The clickpath HPC → Jobs opens a window that displays a list of job IDs, along with the scheduler, user, queue, and status of the job.

7.6.2 Queues Display And Handling In Bright View

The clickpath HPC → Job queues displays a list of queues available (figure 7.5).

![Figure 7.5: Workload Manager Queues](image)

Queues can be added or deleted. An Edit button allows existing queue parameters to be set, for example as in figure 7.6.
7.7 Using cmsh With Workload Management

7.7.1 Jobs Display And Handling In cmsh: jobs Mode

Jobs Mode In cmsh: Top Level

At the top level of jobs mode, the administrator can view all jobs regardless of scheduler type with the list command:

Example

[bright82->jobs]% list

<table>
<thead>
<tr>
<th>Type</th>
<th>Job ID</th>
<th>User</th>
<th>Queue</th>
<th>Status</th>
</tr>
</thead>
<tbody>
<tr>
<td>SGEJob</td>
<td>620</td>
<td>maud</td>
<td>all.q</td>
<td>r</td>
</tr>
<tr>
<td>SGEJob</td>
<td>621</td>
<td>maud</td>
<td>qw</td>
<td></td>
</tr>
<tr>
<td>TorqueJob</td>
<td>90.bright82+</td>
<td>maud</td>
<td>hydroq</td>
<td>S</td>
</tr>
</tbody>
</table>

Also within the jobs mode, the hold, release, suspend, resume, show, and remove commands act on jobs when used with a specified scheduler type and job ID. Continuing with the example:

[bright82->jobs]% suspend torque 90.bright82.cm.cluster

Success

[bright82->jobs]% list

<table>
<thead>
<tr>
<th>Type</th>
<th>jobid</th>
<th>User</th>
<th>Queue</th>
<th>Status</th>
</tr>
</thead>
<tbody>
<tr>
<td>SGEJob</td>
<td>620</td>
<td>maud</td>
<td>all.q</td>
<td>r</td>
</tr>
<tr>
<td>SGEJob</td>
<td>621</td>
<td>maud</td>
<td>qw</td>
<td></td>
</tr>
<tr>
<td>TorqueJob</td>
<td>90.bright82+</td>
<td>maud</td>
<td>hydroq</td>
<td>S</td>
</tr>
</tbody>
</table>

While at the jobs mode top level, the suspended job here can be made to resume using suspend’s complementary command—resume. However, resume along with the other commands can also be executed within a scheduler submode, as is shown shortly.

© Bright Computing, Inc.
jobs **Mode In cmsh:** The scheduler **Submode**

Setting the scheduler type sets the scheduler submode, and can be done thus (continuing with the preceding example):

```
[bright82->jobs] % scheduler torque
[bright82->jobs(torque)]%
```

The submode restriction can be unset with: `scheduler ""`.

The top level job mode commands executed within the scheduler submode then only apply to jobs running under that scheduler. The `list` and `resume` commands, for example, then only apply only to jobs running under torque (continuing with the example):

```
[bright82->jobs(torque)] % list; !#no sge jobs listed now - only torque
-------------------------------------------------------------------
Type     Job ID       User    Queue    Status
---------------------------------------------
TorqueJob 90.bright82+ maud    hydroq S
[bright82->jobs(torque)] % resume 90.bright82.cm.cluster; !#torque job
Success
[bright82->jobs(torque)] % list; !#only torque jobs
-------------------------------------------------------------------
Type     Job ID       User    Queue    Status
---------------------------------------------
TorqueJob 90.bright82+ maud    hydroq R
```

jobs **Mode In cmsh:** The **show** Command

The `show` command for a particular scheduler and job lists details of the job. Continuing with the preceding example:

```
[bright82->jobs(torque)] % show 90.bright82.cm.cluster;
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Arguments</td>
<td>-q hydroq /home/maud/sleeper.sh</td>
</tr>
<tr>
<td>Executable</td>
<td></td>
</tr>
<tr>
<td>In queue</td>
<td></td>
</tr>
<tr>
<td>Job ID</td>
<td>90.bright82.cm.cluster</td>
</tr>
<tr>
<td>Job name</td>
<td>sleeper.sh</td>
</tr>
<tr>
<td>Mail list</td>
<td></td>
</tr>
<tr>
<td>Mail options</td>
<td>a</td>
</tr>
<tr>
<td>Maximum wallclock time</td>
<td>02:00:00</td>
</tr>
<tr>
<td>Memory usage</td>
<td>0</td>
</tr>
<tr>
<td>Nodes</td>
<td>node001</td>
</tr>
<tr>
<td>Notify by mail</td>
<td>yes</td>
</tr>
<tr>
<td>Number of processes</td>
<td>1</td>
</tr>
<tr>
<td>Priority</td>
<td>0</td>
</tr>
<tr>
<td>Queue</td>
<td>hydroq</td>
</tr>
<tr>
<td>Run directory</td>
<td>/home/maud</td>
</tr>
<tr>
<td>Running time</td>
<td>809</td>
</tr>
<tr>
<td>Status</td>
<td>R</td>
</tr>
<tr>
<td>Stderr file</td>
<td>bright82.cm.cluster:/home/maud/sleeper.sh.e90</td>
</tr>
<tr>
<td>Stdout file</td>
<td>bright82.cm.cluster:/home/maud/sleeper.sh.o90</td>
</tr>
<tr>
<td>Submission time</td>
<td>Fri Feb 18 12:49:01 2011</td>
</tr>
<tr>
<td>Type</td>
<td>TorqueJob</td>
</tr>
<tr>
<td>User</td>
<td>maud</td>
</tr>
</tbody>
</table>

jobs **Mode In cmsh:** The **list** Command

The `list` command lists the currently pending and running tasks, and recently completed tasks. For example, on a cluster with nodes `node001` to `node004` there may be a job ID of 27 that is running. The output of `list` might then be:
Example

[bright82->jobs(slurm)]% list

<table>
<thead>
<tr>
<th>Type</th>
<th>Job ID</th>
<th>User</th>
<th>Queue</th>
<th>Running time</th>
<th>Status</th>
<th>Nodes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slurm</td>
<td>25</td>
<td>fred</td>
<td>defq</td>
<td>2s</td>
<td>COMPLETED</td>
<td>node001..node004</td>
</tr>
<tr>
<td>Slurm</td>
<td>26</td>
<td>fred</td>
<td>defq</td>
<td>22s</td>
<td>COMPLETED</td>
<td>node001..node004</td>
</tr>
<tr>
<td>Slurm</td>
<td>27</td>
<td>fred</td>
<td>defq</td>
<td>10s</td>
<td>RUNNING</td>
<td>node001..node004</td>
</tr>
<tr>
<td>Slurm</td>
<td>28</td>
<td>fred</td>
<td>defq</td>
<td>0s</td>
<td>PENDING</td>
<td></td>
</tr>
<tr>
<td>Slurm</td>
<td>29</td>
<td>fred</td>
<td>defq</td>
<td>0s</td>
<td>PENDING</td>
<td></td>
</tr>
</tbody>
</table>

Example

jobs Mode in cmsh: The filter Command
The filter command lists all jobs, and offers ways to filter the displayed output.

The filter command without any options is a bit like an extension of the list command in that it lists currently pending and running jobs, although in a different format. However, in addition, it also lists past tasks, with their start and end times.

An example of the output format, somewhat simplified for clarity, is:

Example

[bright82->jobs(slurm)]% filter

<table>
<thead>
<tr>
<th>Job ID</th>
<th>Job name</th>
<th>User</th>
<th>Queue</th>
<th>Submit time</th>
<th>Start time</th>
<th>End time</th>
<th>Nodes</th>
<th>Exit code</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>hello</td>
<td>fred</td>
<td>defq</td>
<td>15:00:51</td>
<td>15:00:51</td>
<td>15:00:52</td>
<td>node001</td>
<td>0</td>
</tr>
<tr>
<td>3</td>
<td>mpirun</td>
<td>fred</td>
<td>defq</td>
<td>15:03:17</td>
<td>15:03:17</td>
<td>15:03:18</td>
<td>node001</td>
<td>0</td>
</tr>
<tr>
<td>...</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>slurmhello.sh</td>
<td>fred</td>
<td>defq</td>
<td>16:17:30</td>
<td>16:17:31</td>
<td>16:17:33</td>
<td>node001..node004</td>
<td>0</td>
</tr>
<tr>
<td>26</td>
<td>slurmhello.sh</td>
<td>fred</td>
<td>defq</td>
<td>16:18:39</td>
<td>16:18:40</td>
<td>16:19:02</td>
<td>node001..node004</td>
<td>0</td>
</tr>
<tr>
<td>27</td>
<td>slurmhello.sh</td>
<td>fred</td>
<td>defq</td>
<td>16:18:41</td>
<td>16:19:03</td>
<td>16:19:25</td>
<td>node001..node004</td>
<td>0</td>
</tr>
<tr>
<td>28</td>
<td>slurmhello.sh</td>
<td>fred</td>
<td>defq</td>
<td>16:18:41</td>
<td>N/A</td>
<td>N/A</td>
<td>node001..node004</td>
<td>0</td>
</tr>
<tr>
<td>29</td>
<td>slurmhello.sh</td>
<td>fred</td>
<td>defq</td>
<td>16:18:42</td>
<td>N/A</td>
<td>N/A</td>
<td>node001..node004</td>
<td>0</td>
</tr>
</tbody>
</table>

In the preceding example, the start and end times for jobs 28 and 29 are not yet available because the jobs are still pending.

Because all jobs—historic and present—are displayed by the filter command, it means that finding a particular job in the output displayed can be hard.

The following options to the filter command are therefore made available in order to find a particular job:

- --pending: Display only all pending jobs.
- --running: Display only all running jobs.
- --ended: Display only all jobs that ended successfully.
- --failed: Display only all jobs that failed.
- -u|--user: Display only jobs for a specified user user.

It may take about a minute for CMDaemon to become aware of job data. This means that, for example, a job submitted 10s ago may not show up in the output.

Further options to filter can be seen by running the command help filter in jobs mode.
The statistics command without any options displays an overview of states for all past and present jobs:

**Example**

```
[bright82->jobs]% statistics
```

<table>
<thead>
<tr>
<th>Interval</th>
<th>User</th>
<th>Group</th>
<th>Account</th>
<th>Pending</th>
<th>Running</th>
<th>Finished</th>
<th>Error</th>
<th>Nodes</th>
</tr>
</thead>
<tbody>
<tr>
<td>N/A</td>
<td>0</td>
<td>1</td>
<td>20</td>
<td>0</td>
<td>13</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The jobs statistics can be split across users. An example of the output format in this case, somewhat simplified for clarity, is:

**Example**

```
[bright82->jobs(slurm)]% statistics --user
```

<table>
<thead>
<tr>
<th>User</th>
<th>Pending</th>
<th>Running</th>
<th>Finished</th>
<th>Error</th>
<th>Nodes</th>
</tr>
</thead>
<tbody>
<tr>
<td>alice</td>
<td>0</td>
<td>0</td>
<td>2</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>bob</td>
<td>0</td>
<td>0</td>
<td>5</td>
<td>0</td>
<td>4</td>
</tr>
<tr>
<td>charline</td>
<td>0</td>
<td>1</td>
<td>2</td>
<td>0</td>
<td>2</td>
</tr>
<tr>
<td>dennis</td>
<td>0</td>
<td>0</td>
<td>6</td>
<td>0</td>
<td>4</td>
</tr>
<tr>
<td>eve</td>
<td>0</td>
<td>0</td>
<td>4</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>frank</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
</tr>
</tbody>
</table>

The job statistics can be displayed over various time periods, if there are jobs within the period. An example of the output format, somewhat simplified for clarity, for an interval of 60s is:

**Example**

```
[bright82->jobs(slurm)]% statistics --interval 60
```

<table>
<thead>
<tr>
<th>Interval</th>
<th>Pending</th>
<th>Running</th>
<th>Finished</th>
<th>Error</th>
<th>Nodes</th>
</tr>
</thead>
<tbody>
<tr>
<td>15:00:00 - 15:01:00</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>15:03:00 - 15:04:00</td>
<td>0</td>
<td>0</td>
<td>7</td>
<td>0</td>
<td>7</td>
</tr>
<tr>
<td>15:04:00 - 15:05:00</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>1</td>
<td>2</td>
</tr>
<tr>
<td>15:06:00 - 15:07:00</td>
<td>0</td>
<td>0</td>
<td>3</td>
<td>0</td>
<td>7</td>
</tr>
<tr>
<td>15:07:00 - 15:08:00</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>15:08:00 - 15:09:00</td>
<td>0</td>
<td>0</td>
<td>6</td>
<td>0</td>
<td>8</td>
</tr>
<tr>
<td>16:17:00 - 16:18:00</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>4</td>
</tr>
<tr>
<td>16:19:00 - 16:20:00</td>
<td>0</td>
<td>0</td>
<td>9</td>
<td>0</td>
<td>36</td>
</tr>
<tr>
<td>16:20:00 - 16:21:00</td>
<td>0</td>
<td>0</td>
<td>6</td>
<td>0</td>
<td>24</td>
</tr>
<tr>
<td>16:21:00 - 16:22:00</td>
<td>0</td>
<td>0</td>
<td>3</td>
<td>0</td>
<td>12</td>
</tr>
</tbody>
</table>

**Job Directives**

Job directives configure some of the ways in which CMDaemon manages job information processing. The administrator can configure the following directives:

- **JobInformationKeepCount**: The maximal number of jobs that are kept in the cache, default 8192, maximal value 100 million (page 659).
- **JobInformationKeepDuration**: How long to keep jobs in the cache, default 28 days (page 659).
- **JobInformationMinimalJobDuration**: Minimal duration for jobs to place them in the cache, default 0s (page 659).
- **JobInformationFlushInterval**: Over what time period to flush the cache to storage (page 660).
- **JobInformationDisabled**: Disables job information processing (page 659).
- **CacheJobsTime**: Job information is removed from memory after that time (page 659).
7.7.2 Job Queues Display And Handling In cmsh: jobqueue Mode

Properties of scheduler job queues can be viewed and set in jobqueue mode.

jobqueue Mode In cmsh: Top Level

If a scheduler submode is not set, then the list, qstat, and listpes commands operate, as is expected, on all queues for all schedulers.

At the top level of jobqueue mode:

- list lists the queues associated with a scheduler.

Example

```bash
[root@bright82 ~]# cmsh
[bright82]% jobqueue
[bright82->jobqueue]% list
                Type   Name
------------ ------------------------
          sge      all.q
          torque    default
          torque    hydroq
          torque    longq
          torque    shortq
```

- qstat lists statistics for the queues associated with a scheduler.

Example

```bash
[bright82->jobqueue]% qstat
======================= sge ========================
Queue Load Total Used Available
------------ --------- --------- --------- ---------
    all.q 0.1  1  0  1
====================== torque ======================    
Queue Running Queued Held Waiting
------------ --------- --------- --------- ---------
     default     0     0     0     0
     hydroq      1     0     0     0
     longq      0     0     0     0
     shortq      0     0     0     0
```

- listpes lists the parallel environment available for schedulers

Example

```bash
[bright82->jobqueue]% listpes
Scheduler         Parallel Environment
------------ ------------------------
          sge     make
          sge     mpich
          ...
          sge     openmpi_ib
```

© Bright Computing, Inc.
• scheduler sets the scheduler submode

**Example**

[bright82->jobqueue]% scheduler torque
Working scheduler is torque
[bright82->jobqueue(torque)]%

The submode can be unset using: scheduler ""

**jobqueue Mode In cmsh: The scheduler Submode**

If a scheduler submode is set, then commands under jobqueue mode operate only on the queues for that particular scheduler. For example, within the torque submode of jobqueue mode, the `list` command shows only the queues for torque.

**Example**

[bright82->jobqueue]% list
Type Name
------------ ------------------------
sge all.q
torque default
torque longq
torque shortq
[bright82->jobqueue]% scheduler torque
Working scheduler is torque
[bright82->jobqueue(torque)]% list
Type Name
------------ ------------------------
torque default
torque longq
torque shortq

**jobqueue Mode In cmsh: Other Object Manipulation Commands**

The usual object manipulation commands of section 2.5.3 work at the top level mode as well as in the scheduler submode:

**Example**

[bright82->jobqueue]% list torque
Type Name
------------ ------------------------
torque default
 torque longq
torque shortq
[bright82->jobqueue]% show torque longq
Parameter Value
------------------------------ ------------------------------------------
Maximal runtime 23:59:59
Minimal runtime 00:00:00
Queue type Execution
Routes
Type name torquelongq
 nodes node001.cm.cluster node002.cm.cluster
[bright82->jobqueue]% get torque longq maximalruntime
7.7 Using cmsh With Workload Management

23:59:59
[bright82->jobqueue]%
[bright82->jobqueue]% scheduler torque
Working scheduler is torque
[bright82->jobqueue(torque)]% list
<table>
<thead>
<tr>
<th>Type</th>
<th>Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>torque</td>
<td>default</td>
</tr>
<tr>
<td>torque</td>
<td>longq</td>
</tr>
<tr>
<td>torque</td>
<td>shortq</td>
</tr>
</tbody>
</table>
[bright82->jobqueue(torque)]% show longq
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximal runtime</td>
<td>23:59:59</td>
</tr>
<tr>
<td>Minimal runtime</td>
<td>00:00:00</td>
</tr>
<tr>
<td>Queue type</td>
<td>Execution</td>
</tr>
<tr>
<td>Routes</td>
<td></td>
</tr>
<tr>
<td>Type</td>
<td>torque</td>
</tr>
<tr>
<td>name</td>
<td>longq</td>
</tr>
<tr>
<td>nodes</td>
<td>node001.cm.cluster node002.cm.cluster</td>
</tr>
</tbody>
</table>
[bright82->jobqueue(torque)]% get longq maximalruntime
23:59:59
[bright82->jobqueue(torque)]% use longq
[bright82->jobqueue(torque)->longq]% show
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximal runtime</td>
<td>23:59:59</td>
</tr>
<tr>
<td>Minimal runtime</td>
<td>00:00:00</td>
</tr>
<tr>
<td>Queue type</td>
<td>Execution</td>
</tr>
<tr>
<td>Routes</td>
<td></td>
</tr>
<tr>
<td>Type</td>
<td>torque</td>
</tr>
<tr>
<td>name</td>
<td>longq</td>
</tr>
<tr>
<td>nodes</td>
<td>node001.cm.cluster node002.cm.cluster</td>
</tr>
</tbody>
</table>
[bright82->jobqueue(torque)->longq]% get maximalruntime
23:59:59

7.7.3 Nodes Drainage Status And Handling In cmsh

Running the device mode command drainstatus displays if a specified node is in a Drained state or not. In a Drained state jobs are not allowed to start running on that node.

Running the device mode command drain puts a specified node in a Drained state:

Example

[root@bright82 ~]# cmsh
[bright82]% device
[bright82->device]% drainstatus

<table>
<thead>
<tr>
<th>Node</th>
<th>Queue</th>
<th>Status</th>
</tr>
</thead>
<tbody>
<tr>
<td>node001</td>
<td>workq</td>
<td></td>
</tr>
<tr>
<td>node002</td>
<td>workq</td>
<td></td>
</tr>
</tbody>
</table>
[bright82->device]% drain node001
Node | Queue | Status     |
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>node001</td>
<td>workq</td>
<td>Drained</td>
</tr>
</tbody>
</table>

The undrain command unsets the Drained state so that jobs may start running on the node again.

© Bright Computing, Inc.
The drain, undrain, and drainstatus commands have the same grouping options. The grouping options can make the command apply to not just one node, but to a list of nodes, a group of nodes, a category of nodes, a rack, a chassis, an overlay, a role, or a status. Continuing the example:

[bright82->device]% drain -c default; !# for a category of nodes

<table>
<thead>
<tr>
<th>Node</th>
<th>Queue</th>
<th>Status</th>
</tr>
</thead>
<tbody>
<tr>
<td>node001</td>
<td>workq</td>
<td>Drained</td>
</tr>
<tr>
<td>node002</td>
<td>workq</td>
<td>Drained</td>
</tr>
</tbody>
</table>

The help text for each command indicates the syntax:

**Example**

[root@bright82 ~]# cmsh -c "device help drain"

Name: drain - Drain jobs (not data) on a set of nodes

Usage: drain [OPTIONS/node]

Options: 
- -nodes node(list)
  List of nodes, e.g.
  node001..node015,node020..node028,node030 or
  ~/some/file/containing/hostnames

- -group group(list)
  Include all nodes that belong to the node group, e.g.
  testnodes or test01,test03

- -category category(list)
  Include all nodes that belong to the category, e.g. default
  or default,gpu

- -rack rack(list)
  Include all nodes that are located in the given rack, e.g
  rack01 or rack01..rack04

- -chassis chassis(list)
  Include all nodes that are located in the given chassis, e.g
  chassis01 or chassis03..chassis05

- -overlay overlay(list)
  Include all nodes that are part of the given overlay, e.g
  overlay1 or overlayA,overlayC

- -image <image>
  Include all nodes that have the given image, e.g default-image
  or default-image,gpu-image

- -type <type>
  Type of devices, e.g node or virtualnode,cloudnode

- -intersection
  Calculate the intersection of the above selections

- -union
  Calculate the union of the above selections

© Bright Computing, Inc.
7.7 Using cmsh With Workload Management

-\l, --role role
  Filter all nodes that have the given role

-\s, --status <status>
  Only run command on nodes with specified status, e.g. UP, "CLOSED|DOWN", "INST.*"

--setactions <actions>
  set drain actions, actions already set will be removed (comma-separated)

--appendactions <actions>
  append drain actions to already existing drain actions (comma-separated)

--removeactions <actions>
  remove drain actions

--listactions
  list all drain actions

--clearactions
  remove all drain actions

Examples:

- `drain` Drain the current node
- `drain node001` Drain node001
- `drain -r rack01` Drain all nodes in rack01
- `drain -a reboot` Drain the current node, and append reboot when all jobs are completed

A useful one-liner to reboot and reprovision a sequence of nodes would be in this format:

Example

```
cmsh -c 'device; drain -n <nodes> -t "powerreset"; undrain -n <nodes>'
```

This drains the nodes, then triggers the power reset action, which provisions the nodes, and then undrains the nodes again so that they are ready to accept jobs again. The command allows the sequence of nodes to be rebooted, for example for the purpose of upgrading a kernel, without needing to schedule a downtime for the cluster.

7.7.4 Launching Jobs With cm-launcher

Some MPI distributions occasionally leave processes behind that cannot be killed from the workload manager. To prevent this situation from occurring, Bright Cluster Manager provides the cm-launcher wrapper for the mpirun command, which tracks and kills such processes. The tracking relies on knowing what processes the workload manager launches, so it can only run from within a suitable workload manager. Currently, SGE is a suitable workload manager.

In the following job script examples, instead of having users use:

Example

```
mpirun <...>
```

which may not have a job clean up properly after it ends, users can use:

© Bright Computing, Inc.
Example

```bash
cm-launcher mpirun <...>
```

which cleans up properly after the job ends. Here, `<...>` is used to indicate the mix of options, programs and arguments used with `mpirun`.

For SGE the procedure is as follows:

- A symlink to `cm-launcher` is created to the `<arch>` SGE functions directory library
  ```bash
  ln -s /cm/shared/apps/sge/var/cm/cm-launcher /cm/shared/apps/sge/current/bin/<arch>
  ```

- SGE’s `epilog` (spelled without the “-ue” ending) script is set either for a particular queue, or globally.
  - To set it for a particular queue, for example, for the default queue `all.q`, the following `cmsh` commands can be run:

      Example

      ```bash
      [root@bright82 ~]# cmsh
      [bright82]% jobqueue
      [bright82->jobqueue]% scheduler sge
      Working scheduler is sge
      [bright82->jobqueue(sge)]% set all.q epilog /cm/shared/apps/sge/var/cm/cmepilog
      [bright82->jobqueue(sge)]% commit
      Successfully committed 1 JobQueues
      [bright82->jobqueue(sge)]%
      ```

- To set it globally for all queues, that is, not just the queue `all.q` but all the other queues as well, the following SGE configuration command is used:

      ```bash
      qconf -mconf global
      ```

      This starts up an editor acting on the global configuration setting. The `epilog` line entry is modified to:

      ```bash
      epilog /cm/shared/apps/sge/var/cm/epilog
      ```

7.8 Examples Of Workload Management Assignment

7.8.1 Setting Up A New Category And A New Queue For It

Suppose a new node with processor optimized to handle Shor’s algorithm is added to a cluster that originally has no such nodes. This merits a new category, `shornodes`, so that administrators can configure more such new nodes efficiently. It also merits a new queue, `shorq`, so that users are aware that they can submit suitably optimized jobs to this category.

Ways to configure the cluster manager for this are described next.

A New Category And A New Queue With Bright View

To create a new queue, the clickpath `HPC→Job queues` is selected, and the `Add` button is used to add a particular workload manager job queue scheduler, for example `PBS pro job queues`. The queue options must be configured—at least given a name—from the dialog window (figure 7.7).
The queue option configuration is saved, and then the job queue scheduler configuration is saved. Next, a new category can be added via the clickpath Grouping → Node categories → Add. Parameters in the new category can be set—at least the category name should be set—to suit the new machines (figure 7.8).

The name shornodes can therefore be set here.

Another option within the category is to set the queue. The queue, shorq, is therefore set here for this new category. Setting a queue for the category means configuring the options of the queue scheduler role, for example PBS Pro client role, for the category. Continuing from within the Node categories options window of figure 7.8, the relative clickpath to set a queue is Roles → Add → PBS pro client role → Edit. From the queue scheduler options settings window that opens up, the appropriate queues can be selected via a checkbox tick (figure 7.9).
In this case, the shorq created earlier on in figure 7.7 is presented for selection. After ticking the checkbox, the configuration settings can then be saved—for this example it means going back up through the levels and clicking on the Save button at the node category settings level (the right hand window shown in figure 7.8), and also going up one more level to click the Save button on the node category list window (the left hand window shown in figure 7.8).

Nodes that are to use the queue should be members of the shornodes category. The final step is then to allocate such nodes to the category. This can be done, for example for a node001, by going into the settings of the node, via the clickpath Devices→Nodes[node001]→Edit→Settings→Category and setting the category to shornodes.

**A New Category And A New Queue With cmsh**

The preceding example can also be configured in cmsh as follows:

The new queue can be added from within jobqueue mode, for the workload manager. For example, if Slurm is enabled:

```
[bright82% jobqueue add shorq
[bright82->jobqueue*(slurm)->shorq*]% commit
```

The new category, shornodes, can be created by cloning an old category, such as default:

```
[bright82->jobqueue(slurm)->shorq% category
[bright82->category]% clone default shornodes
[bright82->category*shornodes%]% commit
```

Then, going into the roles submode, appropriate workload manager roles can be assigned, and appropriate queues can be appended and committed for that category:

```
[bright82->category*shornodes%]% roles
[bright82->category*shornodes%]->roles% assign slurmclient; commit
[bright82->category*shornodes%]->roles[slurmclient]% append queues shorq
[bright82->category*shornodes%]->roles% commit
```

The nodes belonging to the shornodes category can then be placed by going into device mode to select the nodes to be placed in that category. For example:

```
[bright82->category*shornodes%]->roles% device use node002
[bright82->device[node002%]]% set category shornodes
[bright82->device*node002%]% commit
```
7.8.2 Setting Up A Prejob Check

How It Works

Measurables such as health checks (section 12.2.4) by default run as scheduled tasks over regular intervals. They can optionally be configured to run on demand instead, or as prejob checks, that is, before a job is run.

If a health check is configured with the prejob setting, then its response means the same as that of a health check, that is:

- If the response to a prejob health check is PASS, then it shows that the node is displaying healthy behavior for that particular health check.
- If the response to a prejob health check is FAIL, then it implies that the node is unhealthy, at least for that particular health check.

The reason a prejob health check is treated more carefully is that, if the node it has just run on is unhealthy, then it means that a job submitted to the node may fail, may not be able to start, or may even vanish outright. The way it can vanish in some cases, without any information beyond the job submission “event horizon” leads to this behavior sometimes being called the Black Hole Node Syndrome.

It can be troublesome for a system administrator to pinpoint the reason for such job failures, since a node may only fail under certain conditions that are hard to reproduce later on. It is therefore a good policy to disallow passing a job to a node which has just been flagged as unhealthy by a health check. Thus, a sensible action (section 12.2.6) taken by a prejob health check on receiving a FAIL response would be to put the node in a Drained state (section 7.7.3). The drain state means that Bright Cluster Manager arranges a rescheduling of the job so that the job runs only on nodes that are believed to be healthy. In hindsight it should also be obvious that a node in a drain state can still accept jobs, if the jobs are run from outside of CMDaemon control. Thus, for example if jobs are submitted via workload managers that are not integrated with CMDaemon, then such jobs are not stopped from running on the node.

A node that has been put in a Drained state with a health check is not automatically undrained. The administrator must clear such a state manually.

The failed prejob health check (page 719) is enabled by default, and logs any prejob health check passes and failures. By default there are no prejob health checks configured, so by default this health check should always pass.

Configuration Using Bright View

To configure the monitoring of nodes with a prejob check in Bright View, the clickpath:

Monitoring Configuration → Data Producers → Monitoring data producers → Edit → When

is used to set the When value for when it runs. In this case, When is set to Pre job.

Configuration Using cmsh

Within cmsh, the equivalent can be carried out as follows:

Example

For example, for the ib health check:

```
[bright82->monitoring->setup]% get ib when
Timed
[bright82->monitoring->setup]% set ib when <TAB><TAB>
on\ demand \ pre \ job \ timed
[bright82->monitoring->setup]% set ib when pre \ job
[bright82->monitoring->setup]% commit
```

Configuration Outside Of CMDaemon

For more unusual prejob checking requirements, further details on how prologs are configured are given in section 7.3.3.

© Bright Computing, Inc.
7.9 Power Saving Features

This section discusses power saving options for workload managers:

- a mechanism specifically for Slurm (section 7.9.1)
- a more general mechanism, cm-scale for nodes in a cluster managed by Bright Cluster Manager (Chapter 8)

7.9.1 Slurm

Slurm provides a power saving mechanism that allows nodes in a client role to go into a power save mode. Power saving is disallowed by default for the head node. If power saving is allowed, then it must be enabled on the nodes where it is to function.

Allowing And Enabling Power Saving On Slurm

Power saving can be allowed and enabled at the category or node level as follows:

Using Bright View: If using Bright View, then the clickpaths are as follows:

For individual nodes, for example a node node001, it is:

- Devices → Nodes[node001] → Edit → Settings → Roles → Slurm client role → Power Saving Allowed (figure 7.10)

Using cmsh: If using cmsh, then within category or device mode, the slurmclient role is chosen from the roles submode.

- To allow power saving, the value for powersavingallowed is set to yes within the slurmclient role, and the setting is committed.

Example

[root@bright82 ~]# cmsh
[bright82] % device roles node001
[bright82->device[node001]->roles] % set slurmclient powersavingallowed yes; commit

© Bright Computing, Inc.
7.10 Cgroups

- To enable power saving, the powersavingenabled parameter can be set to yes in the slurmserver role, as shown in the following example:

Example

[root@bright82 ~]# cmsh
[bright82]# device roles bright82
[bright82->device[bright82]->roles]% set slurmserver powersavingenabled yes; commit

The slurmserver role is typically assigned to the head node.

Additional power saving configuration file parameters are described in the Slurm power saving guide at http://slurm.schedmd.com/power_save.html

7.9.2 The cm-scale Service

The cm-scale service can be used by an administrator to reduce the energy and storage costs of compute nodes by changing their power state, or their existence state, according to workload demands. That is, cm-scale automatically scales a cluster up or down, on-demand, by powering up physical nodes, cloud nodes, or virtual nodes. The scaling is carried out according to settings in the ScaleServer role which set the nodes that are to use the cm-scale service.

The cm-scale service is covered extensively in Chapter 8.

7.10 Cgroups

Linux system processes and all their future children can be aggregated into sets. These sets can be made into hierarchical groups with specialized behavior using the Control Groups (cgroups) mechanism. The behaviour is controlled by different subsystems that are attached to the cgroup. A subsystem may, for example, allow particular CPU cores to be allocated, or it may restrict memory, or it may reduce swap usage by processes that belong to the group, and so on.

Details about Linux cgroups and their subsystems can be found at https://www.kernel.org/doc/Documentation/cgroups/cgroups.txt.

As far as workload management is concerned, it makes sense to distinguish between workload manager cgroup settings and system-wide cgroup parameters. The workload manager cgroup settings allow the administrator to configure a workload manager to use cgroups in a particular way, whereas the system-wide cgroup settings allow the administrator to manage cgroups whether a workload manager is used or not.

7.10.1 Cgroups Settings For Workload Managers

If the workload manager allows cgroups usage, then Bright Cluster Manager provides capabilities to manage the cgroup parameters within the workload manager.

Slurm

Slurm supports 3 cgroups-related plugins. These are all enabled by default, and are:

1. proctrack/cgroup: enables process tracking and suspend/resume capability using cgroups. This plugin is more reliable for tracking and control than the former proctrack/linux.

2. task/cgroup: provides the ability to
   - confine jobs and steps to their allocated cpuset
   - bind tasks to sockets, cores and threads
   - confine jobs and steps to specific memory resources and gres devices
3. `jobacct_gather/cgroup`: collects accounting statistics for jobs, steps and tasks using the `cpuacct`, `memory`, and `blkio` cgroups subsystems.

Slurm uses 2 configuration files to store the parameters and devices used for cgroups support:

1. `/etc/slurm/cgroup.conf`: defines parameters used by Slurm's Linux cgroup-related plugins. The file contains a section that is autogenerated by CMDaemon, with cgroups-related parameters defined in the SlurmServer role.

2. `/etc/slurm/cgroup_allowed_devices_file.conf`: declares devices that need to be allowed by default for all jobs. The syntax of this file accepts one device per line. It permits wildcard definitions such as `/dev/sda*` or `/dev/cpu/*/`. The path to this file can be changed in `cmsh` by setting the value of the parameter `Allowed Devices File` within the `cgroups` submode of the `SlurmServer` role for the device.

**Example**

```
[root@bright82 ~]# cmsh
[bright82]% device use bright82; roles; use slurmserver
[bright82->device[bright82]->roles[slurmserver]]% cgroups; get alloweddevicesfile
/etc/slurm/cgroup_allowed_devices_file.conf
```

For Slurm, the administrator can manage cgroups parameters using `cmsh` by going into the `cgroups` submode of the `SlurmServer` role. Parameters that can be managed include:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Configuration Parameter In cgroup.conf</th>
</tr>
</thead>
<tbody>
<tr>
<td>Auto Mount*</td>
<td>Force Slurm to mount cgroup subsystems if they are not mounted yet</td>
<td>CgroupAutomount</td>
</tr>
<tr>
<td>Mount Point</td>
<td>Where cgroup root is mounted</td>
<td>CgroupMountpoint</td>
</tr>
<tr>
<td>Task Affinity*</td>
<td>Set a default task affinity to bind each step task to a subset of the allocated cores using <code>sched_setaffinity</code></td>
<td>TaskAffinity</td>
</tr>
<tr>
<td>Release Agent Dir</td>
<td>Directory containing Slurm cgroup release_agent files</td>
<td>CgroupReleaseAgentDir</td>
</tr>
<tr>
<td>Allowed Devices File</td>
<td>File used to declare the default devices for all the jobs, if <code>ConstrainDevices</code> is true</td>
<td>AllowedDevicesFile</td>
</tr>
</tbody>
</table>

...continues
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Configuration Parameter</th>
<th>In cgroup.conf</th>
</tr>
</thead>
<tbody>
<tr>
<td>Constrain Cores</td>
<td>Constrain allowed cores to the subset of allocated resources</td>
<td>ConstrainCores</td>
<td></td>
</tr>
<tr>
<td>Constrain RAM Space</td>
<td>Constrain the job’s RAM usage</td>
<td>ConstrainRAMSpace</td>
<td></td>
</tr>
<tr>
<td>Constrain Swap Space</td>
<td>Constrain the job’s swap space usage</td>
<td>ConstrainSwapSpace</td>
<td></td>
</tr>
<tr>
<td>Constrain Devices</td>
<td>Constrain the job’s allowed devices based on GRES allocated resources</td>
<td>ConstrainDevices</td>
<td></td>
</tr>
<tr>
<td>Allowed RAM Space</td>
<td>Percentage memory (default is 100%) out of the allocated RAM allowed for the job cgroup RAM. If this percentage is exceeded, then the job steps will be killed and a warning message will be written to standard error.</td>
<td>AllowedRAMSpace</td>
<td></td>
</tr>
<tr>
<td>Allowed Swap Space</td>
<td>Percent allocated memory allowed for the job cgroup swap space</td>
<td>AllowedSwapSpace</td>
<td></td>
</tr>
<tr>
<td>Max RAM Percent</td>
<td>Maximum percent of total RAM for a job</td>
<td>MaxRAMPercent</td>
<td></td>
</tr>
<tr>
<td>Max Swap Percent</td>
<td>Maximum percent of total RAM for the amount of RAM+Swap that may be used for a job</td>
<td>MaxSwapPercent</td>
<td></td>
</tr>
<tr>
<td>Min RAM Space</td>
<td>Minimum MB for the memory limits defined by Allowed RAM Space and Allowed Swap Space</td>
<td>MinRAMSpace</td>
<td></td>
</tr>
</tbody>
</table>
Parameter | Description | Configuration Parameter
---|---|---
Auto Mount* | Force UGE to mount a cgroup subsystems if they are not mounted yet | mount
Cpuset | If true, then core binding is done by the cgroup cpuset subsystem | cpuset
Freezer | If true, then it enables the cgroup freezer subsystem for job suspension and resumption | freezer
Freeze PE Tasks | If true and the freezer subsystem is turned on, then the master task is suspended, and all slave tasks of the parallel job are also frozen | freeze_pe_tasks

* Boolean (takes yes or no as a value)

The options are always written in the *cgroup.conf* file. More details on these options can be found in the *man page* `man(5) cgroup.conf`.

**UGE**

Univa Grid Engine allows cgroups settings to be defined at two levels.

- Globally: The parameters can be set globally in the global configuration. Bright Cluster Manager defines cgroups global settings in the *cgroups* submode of the *UGE*Server role.

- Per host: The parameters can be overridden in the host configuration for particular hosts. Bright Cluster Manager defines cgroups host settings in the *cgroups* submode of the *UGE*Client role.

The global and host level cgroups parameters are the same, and are as follows:

...continues
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Configuration Parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>Killing</td>
<td>If true then UGE signals all processes forked/started by the job until all of them are killed</td>
<td>killing</td>
</tr>
<tr>
<td>Forced NUMA</td>
<td>If true then on NUMA machines only local memory (memory in the same NUMA zone) is allowed to be used when the job requested memory allocation with -mbind</td>
<td>forced_numa</td>
</tr>
<tr>
<td>Virtual Memory Limit</td>
<td>Specifies if virtual memory can be limited with cgroups</td>
<td>h_vmem_limit</td>
</tr>
<tr>
<td>Memory Free Hard</td>
<td>If true then kernel ensures that the job does not use more main memory than required</td>
<td>h_mem_free_hard</td>
</tr>
<tr>
<td>Memory Free Soft</td>
<td>If true, and the hard memory limit is turned off, then the requested memory with m_mem_free is a soft limit</td>
<td>h_mem_free_soft</td>
</tr>
<tr>
<td>Min Memory Limit</td>
<td>A host based minimum memory limit, in bytes or values like 10M, 1G</td>
<td>min_memory_limit</td>
</tr>
<tr>
<td>Devices</td>
<td>Controls which devices the job can access. It permits wildcard definitions such as /dev/sda* or /dev/cpu/*/</td>
<td>devices</td>
</tr>
</tbody>
</table>

...continues
The options are updated in UGE only when cgroups are enabled in the UGEServer or UGEClient roles. To enable them, enabled must be set to true within the cgroups submode of the role. By default only global cgroups settings are enabled in UGE.

OGS
The Open Grid Scheduler version currently-supported by Bright Cluster Manager does not support Linux cgroups.

PBS Pro
PBS Professional supports cgroups through a set of special Python hooks. The account manager at Altair must be contacted by the administrator in order to install these hooks.

LSF
LSF allows resource enforcement to be controlled with the Linux cgroup memory and cpuset subsystems. By default, when LSF is set up with wlm-setup, then both subsystems are enabled for LSF jobs. If job processes on a host use more memory than the defined limit, then the job is immediately killed by the Linux cgroup memory subsystem. The cgroups-related configuration options are available in cmsh or Bright View, and can be found in the cgroups submode of the LSFServer role:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Configuration Parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>Resource Enforce</td>
<td>Controls resource enforcement through the Linux cgroups memory and cpuset subsystem, on Linux systems with cgroups support. The resource can be either memory or cpu, or both cpu and memory, in either order (default: memory cpu)</td>
<td>LSB_RESOURCE_ENFORCE</td>
</tr>
<tr>
<td>Process Tracking*</td>
<td>This parameter, when enabled, has LSF track processes based on job control functions such as termination, suspension, resume, and other signals, on Linux systems which support the cgroups freezer subsystem</td>
<td>LSF_PROCESS_TRACKING</td>
</tr>
<tr>
<td>Linux Cgroup Accounting*</td>
<td>When enabled, processes are tracked based on CPU and memory accounting for Linux systems that support cgroup’s memory and cpuacct subsystems</td>
<td>LSF_LINUX_CGROUP_ACCT</td>
</tr>
</tbody>
</table>

* Boolean (takes yes or no as a value)

The options are updated in LSF only when cgroups is enabled in the LSFServer role. To enable cgroups, enabled must be set to true in the cgroups submode of the role. By default cgroups is set to enabled in LSF.

© Bright Computing, Inc.
In order to control cgroups behaviour Bright Cluster Manager introduces a new node role: CgroupSupervisor. The role is used for two purposes:

- to help collect job metrics based on cgroups
- to allow the configuration of permanent cgroups, managed by CMDaemon

When a cgroup is created manually (for example, using the `mkdir` command), then by default it disappears after the node reboots. An administrator may wish to have persistent cgroups. The current method of creating permanent cgroups is using the `cgconfig` service. The service allows the creation of cgroup hierarchies, attaches the cgroup subsystems to the hierarchies, and manages the subsystems and the cgroups.

When the CgroupSupervisor role is assigned to a node or node category, then the `cgconfig` service is started and monitored by CMDaemon on that node or node category. CMDaemon manages the autogenerated section in the main configuration file of the service, `/etc/cgconfig.conf`. This file contains three types of entries—mount, group and template. CMDaemon manages the group and template entries on all Linux systems and in addition manages the mount entries on non-systemd systems.

Each group entry in `/etc/cgconfig.conf` describes one hierarchy of cgroups. A template entry, described further on, can be helpful when cgroup rules are defined. A mount entry defines where cgroup controllers should be mounted.

In the CgroupSupervisor role, the administrator can add a new permanent cgroup within the cgroups submode of the role. Configuration of each cgroup defined in the role is written by CMDaemon to `/etc/cgconfig.conf`, and is managed by the `cgconfig` service. The cgroup object that can be accessed from `cmsh` or Bright View allows a list of cgroup controllers (accessible from the controllers submode in the cgroup object) to be defined. In turn, each controller object in the controllers submode has a list of properties that depend on the controller type. Available controller types are the following:

- `cgroupcontrollerblkio`: blkio controller
- `cgroupcontrollercpuset`: cpuset controller
- `cgroupcontrollerhugetlb`: hugetlb controller
- `cgroupcontrllernetprio`: net_prio controller
- `cgroupcontrollercpu`: cpu controller
- `cgroupcontrollerdevices`: devices controller
- `cgroupcontrollermemory`: memory controller
- `cgroupcontrollerns`: ns (namespaces) controller
- `cgroupcontrollercpuacct`: cpuacct controller
- `cgroupcontrollerfreezer`: freezer controller
- `cgroupcontrllernetcls`: net_cls controller
- `cgroupcontrollerperf`: perf_event controller

In `cmsh`, the permanent (managed) cgroup is created and configured from within the CgroupSupervisor role:

**Example**
Different cgroup controller types have different set of properties. When a new controller is added into the managed cgroup, then the controller name is required, but it can be any string. If the Enabled parameter for a controller is set to false, then cgroup configuration will not include the settings of that controller.

Each Group entry in /etc/cgconfig.conf creates a new cgroups hierarchy, sets permissions for the
7.10 Cgroups

hierarchy and for the tasks virtual files, and configures subsystems attached to the hierarchy. The cgroup that is created in the example above would lead to the following content in /etc/cgconfig.conf:

**Example**

[root@bright82 ~]# cat /etc/cgconfig.conf

# This section of this file was automatically generated by cmd. Do not edit manually!
# BEGIN AUTOGENERATED SECTION -- DO NOT REMOVE

group cg1

    perm

        admin

            uid = root;
            gid = root;

        task

            uid = root;
            gid = root;

    memory

        memory.swappiness="60";
        memory.move_charge_at_immigrate="0";
        memory.oom_control="1";

# END AUTOGENERATED SECTION -- DO NOT REMOVE

[root@bright82 ~]#

For operating systems that run without systemd (RHEL/SL/CentOS < 7.0), a managed cgroup also allows rules to be configured that allow processes to be moved into this cgroup automatically. For this functionality, the cgred service is used. This starts the cgrulesengd daemon. For systemd systems this functionality is not available, because systemd prevents the cgred service from doing its job. At the time of writing (January 2016) systemd does not as yet provide functionality similar to the cgred service.

The cgrulesengd daemon (started by the cgred service) reads the process movement rules from /etc/cgrules.conf, which can be managed by CMDaemon. In order to define a new rule, the administrator can add it in the rules submode in cgroup mode as follows:

**Example**

[bright82->device[bright82]->roles[cgroupsupervisor]->cgroups[cg1]]% rules
[bright82->device[bright82]->roles[cgroupsupervisor]->cgroups[cg1]->rules]% add alice
[bright82->...*[cgroupsupervisor*]->cgroups*[cg1*]->rules*[alice*]]% set controllernames cpuset
[bright82->...*[cgroupsupervisor*]->cgroups*[cg1*]->rules*[alice*]]% show

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Controller Names</td>
<td>cpuset</td>
</tr>
<tr>
<td>User</td>
<td>alice</td>
</tr>
</tbody>
</table>

[bright82->...*[cgroupsupervisor*]->cgroups*[cg1*]->rules*[alice*]]% commit
[bright82->...*[cgroupsupervisor*]->cgroups*[cg1*]->rules*[alice*]]%

The rule name is the name of the user that the rule is applied to. It is possible to use an asterisk to define a rule for all users. The command in the rule can be set in order to apply rules only for processes spawned by that command. For example if Command is set to ftp, then when the user runs the ftp
command, the FTP process is automatically moved to the specified cgroup, and restrictions set for the
cgroup are automatically applied to that FTP process.

The rule created in the preceding example creates the following configuration file for cgred:

**Example**

```
[root@bright82 ~]# cat /etc/cgrules.conf

# This section of this file was automatically generated by cmd. Do not edit manually!
# BEGIN AUTOGENERATED SECTION -- DO NOT REMOVE
alice cpuset cg1
# END AUTOGENERATED SECTION -- DO NOT REMOVE

[root@bright82 ~]#
```

It is also possible to define a template cgroup that is not created in the filesystem until a process of
the specified user is started. There are several variables that can be used for specifying cgroup paths in
such templates:

- `%u` – is replaced with the name of the user who owns the current process. If name resolution fails,
  UID is used instead.
- `%U` – is replaced with the UID of the specified user who owns the current process.
- `%g` – is replaced with the name of the user group that owns the current process, or with the GID if
  name resolution fails.
- `%G` – is replaced with the GID of the cgroup that owns the current process.
- `%p` – is replaced with the name of the current process. PID is used in case of name resolution
  failure.
- `%P` – is replaced with the of the PID of the current processes.

For example if the cgroup name is `users/%u`, then setting a rule for user `alice` in the cgroup creates
a new cgroup `users/alice` and places the new processes of the user into that cgroup automatically.

More information on cgroup templates is available under `man 5 cgconfig.conf` and `man 5
cgred.conf`.

### 7.11 Custom Node Parameters

Bright Cluster Manager 8.2 allows the administrator to specify the most important node parameters
when a node is configured in a workload manager. But sometimes the workload manager allows the
configuration of other advanced or user-defined settings for the nodes. When an administrator clones
a node in Bright Cluster Manager, or creates it from scratch, then those advanced settings may need to
be configured manually per node in the workload manager. Sometimes the nodes are created or cloned
automatically, for example `cm-scale` (Chapter 8), in which case node parameters customization is not
possible without manual intervention in the workload manager configuration.

Since Bright Cluster Manager version 8.2, the administrator can configure the advanced node set-
tings in the node customizations mode of `cmsh` or Bright View. The customizations are available in
the workload manager client roles, so they can be applied at the node, node category or configuration
overlay levels.

In order to configure a custom node parameter for the workload manager, the administrator adds
the node customization entry and sets its value. The entry is enabled by default, using the parameter
enabled. Setting the value to disabled automatically removes the parameters from the node settings of
the workload manager.

For example, in order to add a new customization entry for PBS Pro, the following `cmsh` commands
can be used:
Different workload managers allow different kinds of node settings to be set. For example, for the Grid Engines, the settings are the so-called complex attributes, as defined by the administrator. But for PBS Pro, the node settings are built-in, and so the administrator can only change their values but not their names. So when the administrator configures node customization entries, Bright Cluster Manager configures them differently depending on the workload manager that is used.

Thus if the administrator creates a new customization entry with key name <KEY> and value <VALUE>, then this is applied to the workload managers as follows:

- **Slurm.** The node customization entry is appended to the NodeName line of the particular nodes in slurm.conf in the form <KEY>=<VALUE>. If the entry is removed or disabled in Bright Cluster Manager configuration, then the entry <KEY>=<VALUE> pair is removed from the NodeName line.

- **PBS Pro and PBS Pro CE.** The entry is configured with the qmgr utility, using this command: set node NODENAME <KEY>=<VALUE>. If the entry is removed or disabled, then it is unset in the qmgr.

- **UGE and OGS.** The entry is applied to exechost complex attribute that must already be defined in the configuration. The complex attribute name will be set to <KEY> and its value to <VALUE>. If the entry is removed or disabled, then it is removed from the exechost settings of the node.

- **LSF.** The entry is set in the lsb.hosts file, in the Hosts section. If the <KEY> of the entry is defined as one of the section columns, then the parameter value is replaced by Bright Cluster Manager to <VALUE>. When the customization entry is removed or disabled, then its value is replaced by () in the section.
The \texttt{cm-scale} Service

8.1 Introduction

The \texttt{cm-scale} service can be used by an administrator to reduce the energy and storage costs of compute nodes by changing their power state, or their existence state, according to workload demands. That is, \texttt{cm-scale} automatically scales a cluster up or down, on-demand, by powering up physical nodes, cloud nodes, or virtual nodes. The scaling is carried out according to settings in the \texttt{ScaleServer} role which set the nodes that are to use the \texttt{cm-scale} service.

The \texttt{cm-scale} service runs as a daemon. It collects information about workloads from different workload engines, and it uses knowledge of the nodes in the cluster. In the case of HPC jobs, the daemon also gathers knowledge of the queues that the jobs are to be assigned to\footnote{HPC SGE/UGE jobs have no queue assignment by default, and are therefore ignored by default. Adding a line such as: 
\texttt{-q all.q} 
in the \texttt{var/default/common/sge\_request} file, under \texttt{/cm/shared/apps/sge/} (or \texttt{/cm/shared/apps/uge/}) assigns SGE (or UGE) jobs to the \texttt{all.q} queue by default, so that \texttt{cm-scale} considers them.}, and also gathers knowledge on which of the HPC jobs are requesting exclusive node access.

Based on the workload engine information and queues knowledge, the \texttt{cm-scale} service can clone and start compute nodes when the workloads are ready to start. The service also stops or terminates compute nodes, when no queued or running workloads remain on the managed nodes.

8.1.1 Use Cases

The \texttt{cm-scale} service can be considered as a basis for the construction of different kinds of dynamic data centers. Within such centers, nodes can be automatically re-purposed from one workload engine setup to another, or they can be powered on and off based on the current workload demand.

A few use cases are discussed next, to show how this basis can be built upon:

1. An organization wants to run PBS Pro and Slurm on the same cluster, but how much one workload engine is used relative to the other varies over time. For this case, nodes can be placed in the same pool and shared. When a node finishes an existing job, the \texttt{cm-scale} service can then re-purpose that node from one node category to another if needed, pretty much immediately. The re-purposing decision for the node is based on the jobs situation in the PBS Pro and Slurm queues.

2. An organization would like to use their cluster for both Kubernetes and for Slurm jobs. For this case, the admin adds Kubernetes- and Slurm-related settings to the \texttt{ScaleServer} role. Using these settings, the \texttt{cm-scale} service then switches nodes from one configuration overlay to another. For example, if Slurm jobs are pending and there is a free Kubernetes node, then the node is turned into a Slurm node pretty much immediately.

3. An organization has a small cluster with Slurm installed on it, and would like to be able to run more jobs on it. This can be carried out using Bright Cluster Manager’s Cluster Extension cloud-bursting capability (Chapter 3 of the \textit{Cloudbursting Manual}) to extend the cluster when there are too many jobs for the local cluster. The extension to the cluster is made to a public cloud provider, such
as AWS or Azure. For this case, the users can use the cmsg utility (section 4.3 of the Cloudbursting Manual) to submit their cloud jobs, and Slurm sbatch for their regular jobs. The cm-scale service then tracks the Slurm queues, and decides whether or not new cloud nodes should be added from the cloud as extensions to the local network and queues. When the jobs are finished, then the cloud nodes are terminated automatically. Cluster extension typically takes several minutes from prepared images, and longer if starting up from scratch, but in any case this change takes longer than simple re-purposing does.

4. An organization runs PBS Pro only and would like to power down free, local, physical nodes automatically, and start up such nodes when new jobs come in, if needed. In this case, cm-scale follows the queues and decides to stop or to start the nodes automatically depending on workload demand. Nodes typically power up in several minutes, so this change takes longer than simple re-purposing does.

5. An organization uses OpenStack as a private cloud, and would like to extend Slurm partitions into that OpenStack private cloud. In this case, cm-scale can clone Bright-managed instances in OpenStack (Chapter 5.3 of the OpenStack Deployment Manual), and adds them to Slurm’s network and partitions. Cloning an instance can take several minutes, so that this change takes longer than simple re-purposing does.

8.1.2 Restrictions

1. Two workload engines of the same kind cannot be served by cm-scale on the same Bright Cluster Manager cluster. For example, two Slurm instances cannot be managed, but Slurm and LSF can.

2. When the service considers whether or not the node is suited for the workload, it considers only the number of CPU cores requested by the workload. It does not consider other types of resources. For example, if a workload needs Xeon Phis, then cm-scale does not verify if the node actually includes Xeon Phis.

**Number of CPU cores calculation:** The cm-scale service uses the number of CPU cores on a node in order to map workloads to the node. The calculation of this number is optimized for many node scenarios.

For each node, at the beginning of each cm-scale iteration, the following procedure is followed step-by-step to determine the CPU cores number. The procedure for the current iteration stops when a step yields a non-zero value, otherwise the next step is tried.

(a) If the engine is a workload manager, then the workload manager client role is considered and its slots parameter is used. In the very unusual case of several workload manager roles assigned to the node at the same time, then the minimum number of slots is calculated.

(b) If the node is a cloud node, then its flavor Type parameter in the case of EC2, or VMSize in case of Azure, is used. These parameters are accessible via cmsh, within device mode for the cloud node, within the cloudsettings submode. For EC2 nodes the flavor value (Type) can be set to a long statement such as: "62 EC2 Compute Units (16 virtual cores with 3.00 EC2 Compute Units each)", and for this statement 16 is used as the number of CPU cores. If the flavor is not defined on the node, then its cloud provider is considered.

(c) If a template node is defined in the dynamic node provider, and it is a cloud node, then the template node flavor is used for the CPU core number in the same way as shown in the preceding step.

(d) If the Default Resources parameter (in a resource provider) contains "cpus=N", where N is a number of CPU cores, then N is used.
(e) If the node exists at this moment (for example, it is not just cloned in Bright Cluster Manager), then the CPU cores number is requested from CMDaemon, which collects nodes system information. This is used as one of the last methods because it is slow.

(f) If the node is defined in the ScaleServer role via the dynamic resource provider, then the CPU cores number is retrieved from its template node. This method is as slow as the preceding method. It is therefore not recommended when many nodes are managed by cm-scale, otherwise the iteration can take several minutes. In such a case, setting the slots parameter manually is typically wiser, so that step 1 is the step that decides the CPU cores number.

If cm-scale does not manage to find a value for the node, then it prints a warning in the log file and does not make use of the node during the iteration. It is usually wise for the administrator to set the slots parameter manually for nodes that otherwise persistently stay unused, so that resources are not wasted.

8.1.3 Setup

The cm-scale-related files are placed in a local directory when the administrator installs the cm-scale package. If cm-scale is needed on a non-head node, then the package should be installed in the corresponding software image. The administrator also typically configures the default user environment for users of the service, so that the cm-scale modules environment automatically loads up (section 2.2.3).

The cm-scale service does not require any special setup scripts or wizards to run. In order to start using the service, it is enough to assign the ScaleServer role to the nodes, and to configure the role properly by setting the role attributes. Unassigning the role stops the cm-scale service.

The service writes logs to /var/log/cm-scale.log. Running the cm-scale daemon in the foreground for debugging or demonstration purposes is also possible, using the -f option, and the logs are then duplicated to STDOUT:

Example

root@bright82$ module load cm-scale
root@bright82$ cm-scale -f

[...]

8.1.4 Comparison With cm-scale-cluster

Versions of Bright Cluster Manager prior to 8.0 had the cm-scale-cluster utility, which provided similar functionality to the current cm-scale utility. However, the older utility did not allow workloads (jobs) to be considered for more than one workload manager at a time, and also supported only HPC workload managers as workload engines.

The current cm-scale service on the other hand, allows, for example, Slurm workloads to be considered on the same cluster, and also supports other types of workload engines besides HPC workload managers.

Furthermore, the cm-scale utility does not use a crontab entry. The old cm-scale-cluster utility required cron to run it regularly, because only one iteration of decision-making was performed by it. The new cm-scale utility, on the other hand, runs the decision-making loop internally within the cm-scale daemon. The decision-making interval is set in the ScaleServer role.

8.2 Configuration

8.2.1 The ScaleServer Role

To configure cm-scale, the cluster administrator configures the ScaleServer role. The role is typically assigned to head nodes:

Example
The role is configured by setting values to its settings. There are some advanced settings for less common options:

```
[bright82]->device*[bright82*]->roles*[scaleserver*]]% show
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add services</td>
<td>yes</td>
</tr>
<tr>
<td>Name</td>
<td>scaleserver</td>
</tr>
<tr>
<td>Provisioning associations</td>
<td>&lt;0 internally used&gt;</td>
</tr>
<tr>
<td>Revision</td>
<td></td>
</tr>
<tr>
<td>Type</td>
<td>ScaleServerRole</td>
</tr>
<tr>
<td>Engines</td>
<td>&lt;0 in submode&gt;</td>
</tr>
<tr>
<td>Resource Providers</td>
<td>&lt;0 in submode&gt;</td>
</tr>
<tr>
<td>Dry Run</td>
<td>no</td>
</tr>
<tr>
<td>Debug</td>
<td>no</td>
</tr>
<tr>
<td>Run Interval</td>
<td>120</td>
</tr>
<tr>
<td>Advanced Settings</td>
<td>&lt;submode&gt;</td>
</tr>
</tbody>
</table>

```
[bright82]->device*[bright82*]->roles*[scaleserver*]->advancedsettings*]]% show
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Revision</td>
<td></td>
</tr>
<tr>
<td>Debug2</td>
<td>no</td>
</tr>
<tr>
<td>Max Threads</td>
<td>16</td>
</tr>
<tr>
<td>Power Operation Timeout</td>
<td>30</td>
</tr>
<tr>
<td>Connection Retry Interval</td>
<td>5</td>
</tr>
<tr>
<td>Log File</td>
<td>/var/log/cm-scale.log</td>
</tr>
<tr>
<td>Pin Queues</td>
<td>no</td>
</tr>
<tr>
<td>Mix Locations</td>
<td>yes</td>
</tr>
<tr>
<td>Failed Node Is Healthy</td>
<td>no</td>
</tr>
<tr>
<td>Azure Disk Image Name</td>
<td>images</td>
</tr>
<tr>
<td>Azure Disk Container Name</td>
<td>vhds</td>
</tr>
<tr>
<td>Azure Disk Account Prefix</td>
<td></td>
</tr>
</tbody>
</table>

An overview of the parameters and submodes is given next. An example showing how they can be configured is given afterwards, in section 8.3.

**ScaleServer Role Global Parameters**

The ScaleServer role has the following global parameters for controlling the cm-scale service itself:

- **Dry Run**: If set, then the service runs in dry run mode. In this mode it may claim that actions have been carried out on the nodes that use the cm-scale service, however, no action is actually carried out on nodes. This mode is useful for demonstration or debug purposes.

- **Log File**: path to the log file, with a default path of `/var/log/cm-scale.log`.

- **Run Interval**: interval, in seconds, between cm-scale decision-making.

**ScaleServer Role Submodes**

Within the ScaleServer role are the following three submodes:

- **advancedsettings**: allows some advanced properties to be set for cm-scale, using the parameters displayed on page 304.
8.2 Configuration

- **resourceproviders**: defines the nodes used by cm-scale. More explicitly, this submode is used to define resource provider objects. The resource providers can be added as static or dynamic types, and can then have nodes and settings defined within them. The nodes allocated to these resource provider objects are what provide resources to cm-scale when that resource provider is requested.

- **engines**: define the engines used by cm-scale. This can be an instance of the type hpc, generic, or kubernetes (page 310).
  - **trackers** (within engines submode): define the trackers used by cm-scale (page 311)

The parameters are enforced only when the next decision-making iteration takes place.

### 8.2.2 Resource Providers

The cm-scale service allows nodes to change state according to the workload demand. These managed nodes are defined by the administrator within the resourceproviders submode of ScaleServer. Bright Cluster Manager 8.2 supports two types of resource providers: static and dynamic node providers.

#### Static Node Provider

When managed nodes are well-known and will not be extended or shrunk dynamically, then a static node provider can be used. Specifying settings for the static node provider allows cm-scale to power on, power off, or re-purpose nodes, based on nodegroups or a list of nodes nodes specified with a node list syntax (page 39).

The static node provider supports the following properties:

- **Enabled**: The static node provider is currently enabled.

- **Nodes**: A list of nodes managed by cm-scale. These can be regular local compute nodes (nodes), cluster extension cloud compute nodes (cnodes), and Bright-managed OpenStack virtual compute nodes (vnodes). For the purposes of this section on cm-scale, these compute nodes can conveniently be called nodes, cnodes, and vnodes. Since compute nodes are typically the most common cluster nodes, significant resources can typically be saved by having the cm-scale service decide on whether to bring them up or down according to demand.
  - cnodes and vnodes can be cloned and terminated as needed. Cloning and terminating saves on cloud storage costs associated with keeping virtual machine images.
  - regular local compute nodes can be started and stopped as needed. This reduces power consumption.

- **Nodegroups**: List of nodegroups with nodes to be managed by cm-scale. Node groups are classed into *types*. The class of node group types is independent of the class of node types, and should not be confused with it. Node types can be physicalnode, cloudnodes, gpus, genericdevice, headnode, eternetswitch and others, and are shown in the first column of the output of the default list command in device mode. Node group types currently (May 2017) consist only of Normal and Storage pair. Only node groups of type Normal can currently be managed by cm-scale.

- **Priority**: The provider priority. Nodes in the pool of a provider with a higher priority are used first by workloads. By default a resource provider has a priority value 0. These priority values should not be confused with the fairsharing priorities of page 310.

#### Dynamic Node Provider

When managed nodes can be cloned or removed from the configuration, then a dynamic node provider should be used. A compute node that is managed by cm-scale as a dynamic node provider is configured as a template node within the dynamic submode of the ScaleServer role.

The dynamic node provider supports the following properties:

© Bright Computing, Inc.
• **Template Node**: A node that will be used as a template for cloning other nodes in the pool. The following restrictions apply to the template node:
  
  - A workload manager client role must be assigned with a positive number of slots.
  - New node names should not conflict with the node names of nodes in a nodegroup defined for the queue.
  - A specific template node is restricted to a specific queue.

A template node only has to exist as an object in Bright Cluster Manager, with an associated node image. A template node does not need to be up and running physically in order for it to be used to create clones. Sometimes, however, an administrator may want it to run too, like the other nodes that are based upon it, in which case the **Start Template Node** and **Stop Template Node** values apply:

- **Start Template Node**: The template node specified in the Template Node parameter is also started automatically on demand.

- **Stop Template Node**: The template node specified in the Template Node parameter is also stopped automatically on demand.

- **Never Terminate**: Number of cloud nodes that are never terminated even if no jobs need them. If there are this number or fewer cloud nodes, then cm-scale no longer terminates them. Cloud nodes that cannot be terminated can, however, still be powered off, allowing them to remain configured in Bright Cluster Manager. As an aside, local nodes that are under cm-scale control are powered off automatically when no jobs need them, regardless of the Never Terminate value.

- **Enabled**: Node provider is currently enabled.

- **Priority**: Node provider priority.

- **Node Range**: Range of nodes that can be created and managed by cm-scale.

- **Network Interface**: Which node network interface is changed on cloning (incremented).

- **Remove Nodes**: Should the new node be removed from Bright Cluster Manager when the node terminates? If the node is not going to be terminated, but just stopped, then it is never removed.

- **Leave Failed Nodes**: If nodes are discovered to be in a state of INSTALLER_FAILED or INSTALLER_UNREACHABLE (section 5.5.4) then this setting decides if they can be left alone, so that the administrator can decide what do with them later on.

- **Default Resources**: List of default resources, in format \[name=value\].
  
  - **cpu**: value is the number of CPUs
  - **mem**: value is in bytes

These must be set when no real node instance is associated with a node defined in Bright Cluster Manager.

---

**Extra Nodes Settings For Node Providers**

Both the dynamic and static node providers support extra node settings. If configured, then cm-scale can start the extra nodes before the first workload is started, and can stop them after the last job from the managed queue is finished.

The most common use case scenario for extra nodes in the case of cloud nodes is a cloud director node. The cloud director node provisions cloud compute nodes and performs other management operations in a cloud.
In the case of non-cloud non-head nodes, extra nodes can be, for example, a license server, a provisioning node, or an additional storage node.

The configuration settings include:

- **Extra Nodes**: A list of extra nodes.

- **Extra Node Idle Time**: The maximum time, in seconds, that extra nodes can remain unused. The cm-scale service checks for the existence of queued and active workloads using the extra node, when the time elapsed since the last check reaches Extra Node Idle Time. If there are workloads using the extra node, then the time elapsed is reset to zero and a timestamp is written into the file cm-scale.state under the directory set by the Spool role parameter. The time stamp is used to decide when the next check is to take place. Setting Extra Node Idle Time=0 means the extra node is stopped whenever it is found to be idle, and started again whenever workloads require it, which may result in a lot of stops and starts.

- **Extra Node Start**: Extra node is started by cm-scale before the first compute node is started.

- **Extra Node Stop**: Extra node is stopped by cm-scale after the last compute node stops.

The following table summarizes the default attributes in cmsh for the static and dynamic resource providers, along the path cmsh->device[]->roles->scaleserver->resourceproviders[dynamic/static]:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>static</th>
<th>dynamic</th>
</tr>
</thead>
<tbody>
<tr>
<td>Revision</td>
<td>static</td>
<td>dynamic</td>
</tr>
<tr>
<td>Type</td>
<td>static</td>
<td>dynamic</td>
</tr>
<tr>
<td>Nodes</td>
<td>N/A</td>
<td>N/A</td>
</tr>
<tr>
<td>Template Node</td>
<td>N/A</td>
<td>no</td>
</tr>
<tr>
<td>Start Template Node</td>
<td>N/A</td>
<td>no</td>
</tr>
<tr>
<td>Stop Template Node</td>
<td>N/A</td>
<td>no</td>
</tr>
<tr>
<td>Remove Nodes</td>
<td>N/A</td>
<td>no</td>
</tr>
<tr>
<td>Leave Failed Nodes</td>
<td>N/A</td>
<td>yes</td>
</tr>
<tr>
<td>Never Terminate</td>
<td>N/A</td>
<td>32</td>
</tr>
<tr>
<td>Node Range</td>
<td>N/A</td>
<td></td>
</tr>
<tr>
<td>Network Interface</td>
<td>N/A</td>
<td>tun0</td>
</tr>
<tr>
<td>Enabled</td>
<td>yes</td>
<td>yes</td>
</tr>
<tr>
<td>Nodegroups</td>
<td>N/A</td>
<td></td>
</tr>
<tr>
<td>Priority</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Whole Time</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Stopping Allowance Period</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Extra Nodes</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Extra Node Idle Time</td>
<td>3600</td>
<td>3600</td>
</tr>
<tr>
<td>Extra Node Start</td>
<td>yes</td>
<td>yes</td>
</tr>
<tr>
<td>Extra Node Stop</td>
<td>yes</td>
<td>yes</td>
</tr>
<tr>
<td>Default Resources</td>
<td>cpus=1</td>
<td>cpus=1</td>
</tr>
</tbody>
</table>

In the preceding table, the entry N/A means that the parameter is not available for the corresponding resource provider.

### 8.2.3 Time Quanta Optimization

Time quanta optimization is an additional feature that cm-scale can use for further cost-saving with certain cloud providers.

For instance, Amazon charges per whole unit of time, or time quantum, used per cloud node, even if only a fraction of that unit of time was actually used. The aim of Bright Cluster Manager’s time quanta optimization is to keep a node up as long as possible within the already-paid-for time quantum, but

© Bright Computing, Inc.
without incurring further cloud provider charges for a node that is not currently useful. That is, the aim is to:

- keep a node up if it is running jobs in the cloud
- keep a node up if it is not running jobs in the cloud, if its cloud time has already been paid for, until that cloud time is about to run out
- take a node down if it is not running jobs in the cloud, if its cloud time is about to run out, in order to avoid being charged another unit of cloud time

Time quanta optimization is implemented with some guidance from the administrator for its associated parameters. The following parameters are common for both static and dynamic node resource providers:

- **Whole time.** A compute node running time (in minutes) before it is stopped if no workload requires it. Currently (May 2017) Amazon’s time quantum is 60 minutes. By default, Bright Cluster Manager uses a value of Whole Time=0, which is a special value that means Whole Time is ignored. Ignoring it means that Bright Cluster Manager does no time quanta optimization to try to optimize how costs are minimized, but instead simply takes down nodes when they are no longer running jobs.

- **Stopping Allowance Period.** A time (in minutes) just before the end of the Whole Time period, prior to which all power off (or terminate) operations must be started. The parameter associated with time quanta optimization is the Stopping Allowance Period. This parameter can also be set by the administrator. The Stopping Allowance Period can be understood by considering the last call time period. The last call time period is the period between the last call time, and the time that the next whole-time period starts. If the node is to be stopped before the next whole-time charge is applied, then the last call time period must be at least more than the maximum time period that the node takes to stop. The node stopping period in a cluster involves cleanly stopping many processes, rather than just terminating the node instance, and can therefore take some minutes. The maximum period in minutes allowed for stopping the node can be set by the administrator in the parameter Stopping Allowance Period. By default, Stopping Allowance Period=0. Thus, for nodes that are idling and have no jobs scheduled for them, only if the last call time period is more than Stopping Allowance Period, does cm-scale stop the node.

The preceding parameters are explained next. Figure 8.1 illustrates a time line with the parameters used in time quanta optimization.
The algorithm that \texttt{cm-scale} follows, with and without time quanta optimization, can now be described using the two parameters explained so far:

1. \texttt{cm-scale} as part of its normal working, checks every \texttt{Run Interval} seconds to see if it should start up nodes on demand or shut down idling nodes.

2. If it sees idling nodes, then:
   
   (a) If \texttt{Whole Time} has not been set, or is 0, then there is no time quanta optimization that takes place. The \texttt{cm-scale} service then just goes ahead as part of its normal working, and shuts down nodes that have nothing running on them or nothing about to run on them.
   
   (b) If a non-zero \texttt{Whole Time} has been set, then a time quanta optimization attempt is made. The \texttt{cm-scale} service calculates the time period until the next time quantum from public cloud starts. This time period is the current \textit{closing time period}. Its value changes each time that \texttt{cm-scale} is run. If
      
      • the current closing time period is long enough to let the node stop cleanly before the next time quantum starts, and
      
      • the next closing time period—as calculated by the next \texttt{cm-scale} run but also running within the current time quantum—is not long enough for the node to stop cleanly before the next quantum starts

then the current closing time period starts at a time called the \textit{last call}.

In drinking bars, the last call time by a bartender allows some time for a drinker to place the final orders. This allows a drinker to finish drinking in a civilized manner. The drinker is meant to stop drinking before closing time. If the drinker is still drinking beyond that time, then a vigilant law enforcement officer will fine the bartender.

Similarly, the last call time in a scaling cluster allows some time for a node to place its orders to stop running. It allows the node to finish running cleanly. The node is meant to stop running before the next time quantum starts. If the node is still running beyond that time, then a vigilant cloud provider will charge for the next whole time period.

The last call time is the last time that \texttt{cm-scale} can run during the current whole-time period and still have the node stop cleanly within that current whole-time period, and before the next whole-time period starts. Thus, when \texttt{Whole Time} has been set to a non-zero time:
i. If the node is at the last call time, then the node begins with stopping
ii. If the node is not at the last call time, then the node does not begin with stopping

The algorithm goes back again to step 1.

8.2.4 Fairsharing Priority Calculations And Node Management

At intervals of Run Interval, cm-scale collects workloads using trackers configured in the ScaleServer role, and puts all the workloads in a single internal queue. This queue is then sorted by priorities. The priorities are calculated for each workload using the following fairsharing formula:

\[ p_{ij} = a_i \times k_1 + e_j \times k_2 \]

where:

- \( p_{ij} \) is the global priority for the \( i \)-th workload of the \( j \)-th engine. Its value is used to re-order the queue.
- \( a_i \) is the age of the workload. That is, how long has passed since the \( i \)-th job submission, in seconds. This typically dominates the priority calculation, and makes older workloads become higher priority.
- \( k_1 \) is the age factor. This is agefactor that can be set in the engine submode of the ScaleServer role. Usually it is 1.
- \( e_j \) is the engine priority. This is priority in the engine submode of the ScaleServer role.
- \( k_2 \) is the engine factor. This is enginefactor in the engine submode of the ScaleServer role.

When all the workload priorities are calculated and the queue is re-ordered, then cm-scale starts to find appropriate nodes for workloads, from the top of the queue where the higher priority workloads are, to the bottom. This way a higher priority engine has a greater chance of getting nodes for its workloads than a lower priority engine.

8.2.5 Engines

Each workload engine considered by cm-scale must be configured within the engines submode within the ScaleServer role. Bright Cluster Manager 8.2 supports the following workload engines:

- Slurm
- PBS Professional (community and commercial versions)
- LSF
- UGE
- SGE

Engines can be of three types:

- hpc: for all HPC (High Performance Computing) workload managers
- kubernetes: for Kubernetes
- generic: for a generic type

Common Parameters For The cm-scale Engines

All three engine types have the following parameters and submode in common, although their values may differ:

- Workloads Per Node: The number of workloads that can be scheduled to run on the same node at the same time.
For a Kubernetes engine this parameter restricts only the number of either the Kubernetes controllers, or the individual jobs (without any controller) per node. It does not restrict the total number of pods that can be mapped per node by the cm-scale scheduler.

For example, a Kubernetes job, or Job with a capital ‘J’ in Kubernetes terminology, may consist of many pods. Then, if Workload Per Node is, for example, 2, then only 2 Jobs can be mapped to the node.

The number of pods is also taken into account by cm-scale, but this number is taken from the Kubernetes::Node role, where the Max Pods option can be set. If the role is not assigned to a node, then cm-scale assumes that there is no possibility for any pods to run on the node.

- **Priority**: The engine priority
- **Age Factor**: Fairsharing coefficient for workload age
- **Engine Factor**: Fairsharing coefficient for engine priority
- **Max Nodes**: The maximum number of running nodes allowed
- **Trackers**: Enters the workload trackers submode

**Non-common parameters for the cm-scale engines:**

- For the hpc engine:
  - **Workload Manager**: A workload manager name. Possible options are:
    - lsf
    - pbspro
    - sge
    - uge
    - slurm

- For the kubernetes engine, the following parameters can be set:
  - **Cluster**: These are the Kubernetes clusters for which pods are to be tracked. Bright Cluster Manager allows multiple Kubernetes clusters to run on a single compute cluster. Kubernetes must be already set up before this setting is configured.
  - **CPU Busy Threshold**: The CPU load is a value that can range from 0 to 1. The CPU Busy Threshold value defines if the node is too busy for new pods. Its default value is: 0.9.
  - **Memory Busy Threshold**: The Memory load is a value that can range from 0 to 1. The Memory Busy Threshold defines if the node is too busy for new pods. Its default value is: 0.9.

The CPU and Memory thresholds configured in the Kubernetes engine help cm-scale to decide when more nodes are needed. But cm-scale also retrieves the number of pods that are already running on the node and compares it with the Max Pods parameter that is configured in the Kubernetes::Node role assigned to the node. If the number of running pods is already equal or greater than the value of Max Pods, then (from the cm-scale point of view) the node cannot fit more pods, which means that a new node is needed.

**8.2.6 Trackers**

A workload tracker is a way to specify the workload and its node requirements to cm-scale. For HPC, the tracker may be associated with a specific queue, and cm-scale then tracks the jobs in that queue.

One or more trackers can be named and enabled within the trackers submode, which is located within the engines submode of the ScaleServer role. A queue (for workload managers) can be assigned to each tracker.
Example

There are three types of tracker objects supported in Bright Cluster Manager 8.2:

- **queue**: Used with an HPC type engine, where each workload (job) is associated with a particular queue. The attribute `Type` takes the value `ScaleHpcQueueTracker`, and the attribute `Queue` is set to the queue name for the job.

- **namespace**: Used with a kubernetes type engine.

- **generic**: Used with a generic type engine.

The following settings are common for both types of trackers:

- **Enabled**: Enabled means that workloads from this tracker are considered by `cm-scale`.

- **Assign Category**: A node category name that should be assigned to the managed nodes. When a node is supposed to be used by a workload, then `cm-scale` should assign the node category to that node. If the node is already running, and has no workloads running on it, but its category differs from the category specified for the jobs of the queue, then the node is drained, stopped and restarted on the next decision-making iteration of `cm-scale`, and takes on the assigned category. Further details on this are given in the section on dynamic nodes repurposing, page 325.

- **Primary Overlays**: A list of configuration overlays.

  If a workload is associated with the tracker for which the overlays are specified by `Primary Overlays`, then the Bright Cluster Manager-managed nodes are appended to those configuration overlays by `cm-scale`. This takes place after the node is removed from the previous overlays that it is associated with.

  If the node is already running, but has not yet been appended to the overlays specified for the workloads of the tracker, then the node is restarted when no other workloads run on the node, before going on to run the workloads with the new overlays.

  - When a workload is associated with the tracker that has `Primary Overlays` set, then the pool of `cm-scale`-managed nodes is checked.
    
    The check is to decide on if a node is to be made available for the workload.
    
    If the node is appended to the `Primary Overlays` already and is not running workloads, then `cm-scale` simply hands over a workload from the tracker to run on the node.
    
    If the node is not appended to the `Primary Overlays` already, and is not running workloads, then `cm-scale` prepares and boots the node as follows:
    
    * the node is drained and rebooted if it is up, or
    * the node is undrained and merely booted if it is not up

    The node is removed from any previous overlays that it was with, before booting up, and it is appended to the new overlays of `Primary Overlays`.

- **The threshold settings**:

  - **Queue Length Threshold**: number of pending workloads that triggers cloudbursting.
  
    - **Age Threshold**: workload pending time threshold, in seconds, that triggers cloudbursting for this workload.

The queue length and age thresholds allow the administrator to set when `cm-scale` starts or creates cloudbursting nodes. Both thresholds can be used at the same time, or just one of them can be used and the other can be ignored by setting it to 0.

If the queue length threshold is set, then `cm-scale` ignores pending workloads that are located higher (added later) than the threshold in the managed queue.

© Bright Computing, Inc.
Example

Assuming there are 5 jobs in the queue, with job IDs 1, 2, 3, 4, and 5, where the 1st one is the first in the queue. If the queue length threshold is 3, then only jobs 1, 2 and 3 are taken into account, while jobs 4 and 5 are ignored.

Example

If the age threshold is set to 100, then only workloads older than 100 seconds are taken into account, while younger jobs are ignored.

The queue type tracker has only one parameter specific to the tracker: `queue`. This is set to the workload queue that is being tracked.

Namespace Tracker

The namespace tracker of `cm-scale` is used to track Kubernetes workloads. It tracks Kubernetes jobs (via its Job controllers) and tracks individual pods. It does not start new nodes for pending pods owned by other types of Kubernetes pod controllers, such as ReplicaSet, DaemonSet, and so on. If non-Job controllers are running, then `cm-scale` will not stop or terminate those nodes.

The tracker settings in `cmsh` or Bright View include additional parameters that are in common with other trackers:

1. **Controller Namespace**: Tracks the Kubernetes namespace name. Only Kubernetes workloads from this namespace are tracked. To track more than one namespace, one tracker must be created per namespace.

2. **Object**: Type of Kubernetes objects to track. Bright Cluster Manager supports the following object types:
   
   (a) **Job**: A Kubernetes Job controller type represents one or several pods that are expected to eventually terminate. The controller nature makes this type of Kubernetes workload very suited to dynamic data centers.

   (b) **Pod**: Individual pod, without any controller.

If the specified namespace does not exist in Kubernetes, then the tracked jobs or individual pods in this namespace are ignored by `cm-scale`.

Generic Tracker

The `cm-scale` service is able to use a generic tracker to deal with workloads that use various tracker types.

When `cm-scale` starts a new iteration, it re-reads the engines and trackers settings from the ScaleServer role, and searches for the appropriate modules in its directories. In the case of a custom tracker, the module is always loaded according to the tracker handler path. When the tracker module is loaded, `cm-scale` requests a list of workloads from each of the tracker modules. So, the aim of the tracker module is to collect and provide the workloads to `cm-scale` in the correct format.

The path to the tracker module should be specified in the handler parameter of the generic tracker entity using `cmsh` or Bright View as follows:

**Example**

```
[root@bright82 ~]# cmsh
[bright82]# device roles master
[bright82->device[bright82]->roles]# use scaleserver
```
In the preceding example, the handler file `/cm/local/apps/cm-scale/lib/examples/tracker.py` is an example that is provided with the `cm-scale` package. Another example module file provided with the package is `/cm/local/apps/cm-scale/lib/examples/workload.py`, which implements the ExampleWorkload class. Together, the two examples can be used to generate a random number of simple workloads on each `cm-scale` iteration. It is recommended to use the tracker and workload classes as templates for custom tracker and workload modules created by the administrator.

The generic engine does not have any specific parameters associated with its type. It only has parameters common to all of the engine types.

If the generic engine is configured with at least one generic tracker in its role, then `cm-scale` loads the handler module and starts to use the two functions that are implemented in the class. The class name can be chosen arbitrarily:

1. `to_string`: returns a string that identifies the tracker. This is printed to the log file.
2. `get_workloads`: returns a list of objects belonging to a new class inherited from the Workload class. This new class should be created by the administrator.

The workload class must provide the following functions. The class name can be chosen arbitrarily:

1. `__init__`: initializes the workload object.
2. `to_string`: returns a string that identifies the workload. This is printed to the log file.

For example, the following very simple tracker and workload classes can be implemented:

**Example**

```python
class ExampleTracker(Tracker):
    def to_string(self):
        return "my workload"

    def get_workloads(self):
        return [ExampleWorkload(self)]

class ExampleWorkload(Workload):
    def __init__(self, tracker):
        Workload.__init__(self, tracker)
        self.set_id("1")
        self._update_state()
        self._update_age()
        self._update_resources()

    def to_string(self):
        return "workload %s" % self._id

    def _update_state():
        self._set_pending()
```

© Bright Computing, Inc.
def _update_age(self):
    self._age = 0

def _update_resources(self):
    node_res = NodeResource("*")
    cpus_res = CpusResource(1)
    node_res.add_resource(cpu_res)
    engine_res = EngineResource("myengine")
    node_res.add_resource(engine_res)
    self.add_resource(node_res)

The classes should be located in different files. The ExampleWorkload class initializes the workload object with a state, age, and required resources. These values are described below.

State: The state can be pending, running or failed, which can be set with these appropriate functions:

• self._set_pending()
• self._set_running()
• self._set_failed()

If the state is running, then the workload is treated as one that occupies the nodes defined in the resources list. Each NodeResource object in the resources thus represents one occupied node.

If the state is pending, then the workload is treated as one that waits for free nodes. In this case cm-scale tries to find (start or clone) some more nodes in order to allow the workload engine to start this workload.

The failed workload state is considered by cm-scale as exceptional. Such a workload is logged in the log file, but is not considered when cm-scale decides what nodes to start or clone. Any other workload state is also ignored by cm-scale.

Age: The age defines how many seconds the workload is waiting for its resources since being added to the engine. Usually the engine can provide such information, so in the example age is set to 0, which means the workload has been added to the workload engine just now. The age is used in the fairsharing workload priority calculation (page 310). The value of age is not re-calculated by cm-scale after a while. This means that the number that the module sets in the class is used during the iteration, and then forgotten by cm-scale until the workload object is recreated from scratch on the next iteration.

Resources: The Workload class (a base class) includes the resources property with list types. This list includes resource objects that are used by cm-scale in order to find appropriate nodes for the workload. The top level resource type is always NodeResource. There can be one or several node resources requested by the workload.

If the node names are known, then one NodeResource object is created per compute node.

Otherwise a single NodeResource object is used as many times as the number of requested nodes, with the name set to *, which is treated by cm-scale as any suitable node. The number of nodes can be set in the NodeResource object with the set_amount(number) function of the resource.

In the preceding example one (any) node resource is added to the workload request, and the requirement for CPU (cores) number is set to 1. The engine resource is used in order to restrict the mapping of workloads from different engines to one node. Thus if a node has this resource assigned, then the node can take on the workload. If no engine resource is assigned to the node, then it can also take on
The workload, but the engine resource of the workload is assigned to the node before other workloads are considered.

The resource types that can be added to the workload are defined in the Python module `/cm/local/apps/cm-scale/lib/core/resource.py`:

- **NodeResource**: top level resource, contains all other resources.
- **CpusResource**: defines the number of cpu cores required or already used by the workload.
- **CategoryResource**: node category required by the workload.
- **OverlayResource**: required configuration overlay.
- **QueueResource**: HPC queue that the workload (job) belongs to. Used only with engines that support queues.
- **EngineResource**: engine name that the workload belongs to.
- **FeatureResource**: required node feature (node property, in other terminology) that should be supported by the engine.

Custom resource types are not supported for now.

### Enabling Node Shutdown

By default, cm-scale powers off nodes belonging to a resource pool once there is no more workload for them. Resource Providers can also enable shutdown, to give the node time to terminate gracefully.

Shutdown has two options that can be set by the cluster administrator in the configuration file at `/cm/local/apps/cm-scale/lib/config.py`:

- **SHUTDOWN_ENABLED**: If set to `yes`, then the shutdown command is run to terminate the system services first, and after that a command is run to power off the system. A waiting time of `SHUTDOWN_TIMEOUT` seconds takes place between the two events.

- **SHUTDOWN_TIMEOUT**: The number of seconds to wait before powering off a node that is in a shutdown state.

It may take more than `Shutdown Timeout` seconds for a node to power off, depending on the `Run Interval` setting. For example, if `Shutdown Timeout` is 60 seconds, and `Run Interval` is 50, then effectively the `Shutdown Timeout` is 100 seconds, because the power off event only occurs during an iteration execution of cm-scale.

### 8.3 Examples Of cm-scale Use

#### 8.3.1 Simple Static Node Provider Usage Example

The example session that follows explains how a static node provider (page 305) can be configured and used with cm-scale. The session considers a default cluster with a head node and 5 regular nodes which have been previously defined in Bright Cluster Manager configuration. 3 of the regular nodes are powered down at the start of the run. The power control for the nodes must be functioning properly, or otherwise cm-scale cannot power nodes on and off.

The head node has the Slurm server role by default, and the regular nodes run with the Slurm client role by default. So, on a freshly-installed cluster, the `roleoverview` command should show something like:

Example
8.3 Examples Of cm-scale Use

```
[bright82->device[bright82]]% roleoverview | head -2 ; roleoverview | grep slurm

Role  Nodes  Categories  Nodes up
-------------- ----------------- ------------ ---------
slurmclient  node001..node005  default  2 of 5
slurmserver  bright82          1 of 1
```

A test user, fred can be created by the administrator (section 6.2), and an MPI hello executable based on the hello.c code (from section 3.3.3 of the User Manual) can be built:

**Example**

```
[fred@bright82 ~]$ module add shared openmpi/gcc/64/1.10.3 slurm
[fred@bright82 ~]$ mpicc hello.c -o hello
```

A batch file slurmhello.sh (from section 5.3.1 of the User Manual) can be set up. Restricting it to 1 process per node so that it spreads over nodes easier for the purposes of the test can be done with the settings:

**Example**

```
[fred@bright82 ~]$ cat slurmhello.sh
#!/bin/sh
#SBATCH -o my.stdout
#SBATCH --time=30 #time limit to batch job
#SBATCH --ntasks=1
#SBATCH --ntasks-per-node=1
module add shared openmpi/gcc/64/1.10.1 slurm
mpirun /home/fred/hello
```

The user fred can now flood the default queue, defq, with the batch file:

**Example**

```
[fred@bright82 ~]$ while (true); do sbatch slurmhello.sh; done
```

After putting enough jobs into the queue (a few thousand should be enough, and keeping it less than 5000 would be sensible) the flooding can be stopped with a ctrl-c.

The activity in the queue can be watched:

**Example**

```
[root@bright82 ~]# watch "squeue | head -3 ; squeue | tail -3"
```

```
Every 2.0s: squeue | head -3 ; squeue | tail -3  Thu Sep 15 10:33:17 2016

<table>
<thead>
<tr>
<th>JOBID</th>
<th>PARTITION</th>
<th>NAME</th>
<th>USER</th>
<th>ST</th>
<th>TIME</th>
<th>NODES</th>
<th>NODELIST(REASON)</th>
</tr>
</thead>
<tbody>
<tr>
<td>6423</td>
<td>defq</td>
<td>slurmhel</td>
<td>fred</td>
<td>CF</td>
<td>0:00</td>
<td>1</td>
<td>node001</td>
</tr>
<tr>
<td>6424</td>
<td>defq</td>
<td>slurmhel</td>
<td>fred</td>
<td>CF</td>
<td>0:00</td>
<td>1</td>
<td>node002</td>
</tr>
<tr>
<td>6572</td>
<td>defq</td>
<td>slurmhel</td>
<td>fred</td>
<td>PD</td>
<td>0:00</td>
<td>1</td>
<td>(Priority)</td>
</tr>
<tr>
<td>6422</td>
<td>defq</td>
<td>slurmhel</td>
<td>fred</td>
<td>R</td>
<td>0:00</td>
<td>1</td>
<td>node001</td>
</tr>
<tr>
<td>6422</td>
<td>defq</td>
<td>slurmhel</td>
<td>fred</td>
<td>R</td>
<td>0:00</td>
<td>1</td>
<td>node002</td>
</tr>
</tbody>
</table>
```

The preceding indicates that node001 and node002 are being kept busy running the batch jobs, while the remaining nodes are not in use.

The administrator can check on the job status via the job metrics of cmsh too, using the options to the filter command, such as --pending or --running:

© Bright Computing, Inc.
The cm-scale Service

Example

[root@bright82 ~]# cmsh -c "jobs ; watch filter --running -u fred"

Every 2.0s: filter --running -u fred Wed May 7 12:50:05 2017

<table>
<thead>
<tr>
<th>Job ID</th>
<th>Job name</th>
<th>User</th>
<th>Queue</th>
<th>Submit time</th>
<th>Start time</th>
<th>Nodes</th>
<th>Exit code</th>
</tr>
</thead>
<tbody>
<tr>
<td>406</td>
<td>slurmhello.sh</td>
<td>fred</td>
<td>defq</td>
<td>16:16:56</td>
<td>16:27:21</td>
<td>node001,node002</td>
<td>0</td>
</tr>
</tbody>
</table>

and eventually, when jobs are no longer running, it should show something like:

[root@bright82 ~]# cmsh -c "jobs ; watch filter --running -u fred"

Every 2.0s: filter --running Wed May 7 12:56:53 2017

No jobs found

So far, the cluster is queuing or running jobs without cm-scale being used.

The next steps are to modify the behavior by bringing in cm-scale. The administrator assigns the ScaleServer role to the head node. Within the role a new static node provider, Slurm engine, and queue tracker for the defq are set as follows:

Example

[root@bright82 ~]# cmsh

[bright82]% device roles master
[bright82->device[bright82]->roles]% use scaleserver
[bright82->device[bright82]->roles[scaleserver]]% resourceproviders
...->roles[scaleserver]->resourceproviders]% add static pool1
...->roles[scaleserver]->resourceproviders*[pool1*]]% set nodes node001..node005
...->roles[scaleserver]->resourceproviders*[pool1*]]% commit
...->roles[scaleserver]->resourceproviders*[pool1*]
...->roles[scaleserver]%
...->roles[scaleserver]->engines]
...->roles*[scaleserver*]->engines*[slurm1*]]% set workloadmanager slurm
...->roles*[scaleserver*]->engines*[slurm1*]]% trackers
...->roles*[scaleserver*]->engines*[slurm1*]->trackers*]
...->roles*[scaleserver*]->engines*[slurm1*]->trackers*[tr1*]]% set queue defq
...->roles*[scaleserver*]->engines*[slurm1*]->trackers*[tr1*]]% commit
...->roles*[scaleserver*]->engines*[slurm1*]->trackers*[tr1*]]%

The nodes node001..node005 should already be in the queue defq, as assigned to them by default when they were assigned the SlurmClient role. With these settings, they can now be powered up or down on demand by cm-scale service, depending on the number of jobs that are pending. When the new ScaleServer role is committed in cmsh or Bright View, then the cm-scale service is started. If needed, the administrator can check the log file /var/log/cm-scale to see what the service is doing.

On each iteration cm-scale checks whether the node states should be changed. Thus after a while, the nodes node003..node005 are started. Once up, they can start to process the jobs in the queue too.

Watching the running jobs should show something like:

Example

[root@bright82 ~]# cmsh -c "jobs ; watch filter --running"

Every 2.0s: filter --running Thu Sep 8 13:01:59 2016

<table>
<thead>
<tr>
<th>Job ID</th>
<th>Job name</th>
<th>User</th>
<th>Queue</th>
<th>Submit time</th>
<th>Start time</th>
<th>Nodes</th>
<th>Exit code</th>
</tr>
</thead>
<tbody>
<tr>
<td>16498</td>
<td>slurmhello.sh</td>
<td>fred</td>
<td>defq</td>
<td>07/05/2017 12:44:20</td>
<td>07/05/2017 13:20:58</td>
<td>node003</td>
<td>0</td>
</tr>
<tr>
<td>16499</td>
<td>slurmhello.sh</td>
<td>fred</td>
<td>defq</td>
<td>07/05/2017 12:44:20</td>
<td>07/05/2017 13:20:58</td>
<td>node002</td>
<td>0</td>
</tr>
</tbody>
</table>

Eventually, cm-scale finds that all jobs have been dealt with, and the nodes are then powered down.
High-availability And Using A Configuration Overlay For The ScaleServer Role

For high-availability clusters, where there are two head nodes, the scaleserver should run on the active head node. One labor-intensive way to set this up is to assign the service to both the head nodes, and match the scaleserver settings on both head nodes. A simpler way is to define a configuration overlay for the head nodes for the scaleserver. If the head nodes are bright82-1 and bright82-2, then a configuration overlay called brightheads can be created and assigned the service as follows:

Example

[bright82-1]% configurationoverlay add brightheads
[bright82-1->configurationoverlay*brightheads*]% append nodes bright82-1 bright82-2
[bright82-1->configurationoverlay*brightheads*]% roles
[bright82-1->configurationoverlay*brightheads*->roles]% assign scaleserver
[bright82-1->configurationoverlay*brightheads*->roles*[scaleserver*]]%

The scaleserver can then be configured within the configuration overlay instead of on a single head as was done previously in the example of page 318. After carrying out a commit, the scaleserver settings modifications are then mirrored automatically between the two head nodes.

Outside the scaleserver settings, one extra modification is to set the cm-scale service to run on a head node if the head node is active. This can be done with:

Example

[bright82-1->configurationoverlay*[brightheads*]->roles*[scaleserver*]]%

The result is a scaleserver that runs when the head node is active.

8.3.2 Simple Dynamic Node Provider Usage Example

The following example session explains how a dynamic node provider (page 305) can be configured and used with cm-scale. The session considers a default cluster with a head node and 2 regular nodes which have been previously defined in Bright Cluster Manager configuration, and also 1 cloud director node and 2 cloud compute nodes. The cloud nodes can be configured using cm-cluster-extension.

Only the head node is running at the start of the session, while the regular nodes and cloud nodes are all powered down at the start of the run.

At the start, the device status shows something like:

Example

[bright82->device]% ds
eu-west-1-cn001 ....... [  DOWN  ] (Unassigned)
enu-west-1-cn002 ....... [  DOWN  ] (Unassigned)
enu-west-1-cn003 ....... [  DOWN  ] (Unassigned)
enu-west-1-director ....... [  DOWN  ]
n001 .................. [  DOWN  ]
n002 .................. [  DOWN  ]
bright82 ................. [  UP  ]
The power control for the regular nodes must be functioning properly, or otherwise cm-scale cannot power them on and off.

If the head node has the slurmserver role, and the regular nodes have the slurmclient role, then the roleoverview command should show something like:

Example

[bright82->device[bright82]]% roleoverview

<table>
<thead>
<tr>
<th>Role</th>
<th>Nodes</th>
<th>Categories</th>
<th>Nodes up</th>
</tr>
</thead>
<tbody>
<tr>
<td>boot</td>
<td>bright82</td>
<td></td>
<td>1 of 1</td>
</tr>
<tr>
<td>cgroupsupervisor</td>
<td>eu-west-1-cnode001..eu-west-1-cnode002</td>
<td>aws-cloud-director,default,node001..node002</td>
<td>1 of 6</td>
</tr>
<tr>
<td></td>
<td></td>
<td>eu-west-1-cloud-node</td>
<td></td>
</tr>
<tr>
<td>clouddirector</td>
<td>eu-west-1-director</td>
<td></td>
<td>0 of 1</td>
</tr>
<tr>
<td>cloudgateway</td>
<td>bright82</td>
<td></td>
<td>1 of 1</td>
</tr>
<tr>
<td>login</td>
<td>bright82</td>
<td></td>
<td>1 of 1</td>
</tr>
<tr>
<td>master</td>
<td>bright82</td>
<td></td>
<td>1 of 1</td>
</tr>
<tr>
<td>monitoring</td>
<td>eu-west-1-director,bright82</td>
<td></td>
<td>1 of 1</td>
</tr>
<tr>
<td>provisioning</td>
<td>eu-west-1-cnode001..eu-west-1-cnode002</td>
<td>default,node001..node002</td>
<td>0 of 3</td>
</tr>
<tr>
<td>slurmclient</td>
<td>eu-west-1-cnode001..eu-west-1-cnode002</td>
<td>default,node001..node002</td>
<td>0 of 3</td>
</tr>
<tr>
<td>storage</td>
<td>eu-west-1-director,bright82</td>
<td>aws-cloud-director</td>
<td>1 of 1</td>
</tr>
</tbody>
</table>

A test user, fred can be created by the administrator (section 6.2), and an MPI hello executable based on the hello.c code (from section 3.3.3 of the User Manual) can be built:

Example

[fred@bright82 ~]$ module add shared openmpi/gcc/64/1.10.3 slurm
[fred@bright82 ~]$ mpicc hello.c -o hello

A batch file slurmhello.sh (from section 5.3.1 of the User Manual) can be set up. Restricting it to 1 process per node so that it spreads over nodes easier for the purposes of the test can be done with the settings:

Example

[fred@bright82 ~]$ cat slurmhello.sh
#!/bin/sh
#SBATCH -o my.stdout
#SBATCH --time=30 #time limit to batch job
#SBATCH --ntasks=1
#SBATCH --ntasks-per-node=1
module add shared openmpi/gcc/64/1.10.1 slurm
mpirun /home/fred/hello

A default cluster can queue or run jobs without cm-scale being used. The default behavior is modified in the next steps, which bring in the cm-scale service:

The administrator assigns the ScaleServer role to the head node.

Example

[root@bright82 ~]# cmsh
[bright82]% device roles master
[bright82->device[bright82]->roles]% assign scaleserver
Within the assigned scaleserver role, a new dynamic node provider can be set, and properties for the dynamic pool of nodes can be set for the cloud compute nodes. Here the properties that are set are priority (page 305), templatenode (page 306), noderange (page 306), and extranodes (page 306).

Example

```
[bright82->device*[bright82*]->roles*[scaleserver*]]% resourceproviders
    ...->roles*[scaleserver*]->resourceproviders*]% add dynamic pool2
    ...resourceproviders*[pool2*]]% set priority 2
    ...resourceproviders*[pool2*]]% set noderange eu-west-1-cnode001..eu-west-1-cnode002
    ...resourceproviders*[pool2*]]% set templatenode eu-west-1-cnode001
    ...resourceproviders*[pool2*]]% set extranodes eu-west-1-director
    ...resourceproviders*[pool2*]]% commit
    ...resourceproviders*[pool2*]]%
```

The regular compute nodes, node001..node002 should be specified as nodes in the static pool.

The administrator may notice the similarity of dynamic and static pool configuration. The Bright Cluster Manager front end has deliberately been set up to present dynamic pool and static pool nodes to the cluster administrator as two different configuration methods. This is because separating the pool types as dynamic and static pools is simpler for the cluster administrator to deal with. This way, regular compute nodes are treated, not as a special case of a dynamic pool, but simply as static pool nodes. The fundamental reason behind this separate treatment is because physical nodes cannot “materialize” dynamically with properties in the way the cloud compute nodes—which are virtualized nodes—can, due to the need to associate a MAC address with a physical node.

Assigning regular compute nodes to a static pool can be done in a similar way to what was shown before in the example on page 318.

Continuing with the current the session, the nodes node001..node002 are added to the static pool of nodes, on-premise-nodes. For this example they are set to a lower priority than the cloud nodes:

Example

```
...->roles*[scaleserver*]->resourceproviders*]% add static on-premise-nodes
    ...->roles*[scaleserver*]->resourceproviders*[on-premise-nodes*]]% set nodes node001..node002
    ...->roles*[scaleserver*]->resourceproviders*[on-premise-nodes*]]% set priority 1
    ...->roles*[scaleserver*]->resourceproviders*[on-premise-nodes*]]% commit
    ...->roles*[scaleserver*]->resourceproviders*[on-premise-nodes*]]%
```

What this lower priority means is that a node that is not up and is in the static pool of nodes, is only powered on after all the cloud nodes are powered on and busy running jobs. If there happen to be nodes from the static pool that are already up, but are not running jobs, then these nodes take a job, despite the lower priority of the static pool, and irrespective of whether the dynamic pool nodes are in use.

Job priorities can be overridden in cm-scale by:

- allowing locations by setting Mix Locations to true (page 327) or
- pinning queues by setting Pin Queues to true (page 330)

A Slurm engine, and queue tracker for the defq are set as follows:

Example

```
...]->roles*[scaleserver*]]% engines
    ...]->roles*[scaleserver*]->engines*]]% add hpc slurm2
    ...*]->roles*[scaleserver*]->engines*[slurm2*]]% set workloadmanager slurm
    ...*]->roles*[scaleserver*]->engines*[slurm2*]]% set trackers
    ...*]->roles*[scaleserver*]->engines*[slurm2*]->trackers*]]% add queue tr2
    ...*]->roles*[scaleserver*]->engines*[slurm2*]->trackers*[tr2*]]% set queue defq
    ...*]->roles*[scaleserver*]->engines*[slurm*]->trackers*[tr2*]]% commit
    ...]->roles*[scaleserver*]->engines*[slurm1*]->trackers*[tr2*]]%
```

© Bright Computing, Inc.
The nodes node001..node002 and eu-west-1-cnode001..eu-west-1-cnode002 should already be in the queue defq by default, ready to run the jobs:

**Example**

```
...->roles[scaleserver]->engines[slurm1]->trackers[tr2]]% jobqueue use slurm defq ; get nodes
eu-west-1-cnode001
eu-west-1-cnode002
node001
node002
```

The roleoverview (page 320) command is also handy for an overview, and to confirm that the role assignment of these nodes are all set to the SlurmClient role:

With these settings, the nodes in the dynamic pool can now be powered up or down on demand by cm-scale service, depending on the number of jobs that are pending. When the new ScaleServer role is committed in cmsh or Bright View, then the cm-scale is run periodically. Each time it is run, cm-scale checks whether the node states should be changed. If needed, the administrator can check the log file /var/log/cm-scale to see what the service is doing.

Job submission can now be carried out, and the scaleserver assignment carried out earlier scales the cluster to cope with jobs according to the configuration that has been carried out in the session.

Before submitting the batch jobs, the administrator or user can check the jobs that are queued and running with the squeue command. If there are no jobs yet submitted, the output is simply the squeue headers, with no job IDs listed:

**Example**

```
[fred@bright82 ~]$ squeue
JOBID PARTITION NAME USER ST TIME NODES NODELIST(REASON)
```

As in the previous example for the static pool only case (page 316), a way for user fred to flood the default queue defq is to run the batch file in a loop:

**Example**

```
[fred@bright82 ~]$ while (true); do sbatch slurmhello.sh; done
Submitted batch job 1
Submitted batch job 2
Submitted batch job 3
...
```

After putting enough jobs into the queue (a few thousand should be enough, not more than five thousand would be sensible), the flooding can be stopped with a ctrl-c.

The changes in the queue can be watched by user fred:

**Example**

```
[fred@bright82 ~]$ watch "squeue | head -5 ; squeue | tail -4"
Every 2.0s: squeue | head -5 ; squeue | tail -4  Wed Nov 22 16:08:52 2017
```

<table>
<thead>
<tr>
<th>JOBID</th>
<th>PARTITION</th>
<th>NAME</th>
<th>USER</th>
<th>ST</th>
<th>TIME</th>
<th>NODES</th>
<th>NODELIST(REASON)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>defq</td>
<td>slurmhel</td>
<td>fred</td>
<td>PD</td>
<td>0:00</td>
<td></td>
<td>(Resources)</td>
</tr>
<tr>
<td>2</td>
<td>defq</td>
<td>slurmhel</td>
<td>fred</td>
<td>PD</td>
<td>0:00</td>
<td></td>
<td>(Resources)</td>
</tr>
<tr>
<td>3</td>
<td>defq</td>
<td>slurmhel</td>
<td>fred</td>
<td>PD</td>
<td>0:00</td>
<td></td>
<td>(Resources)</td>
</tr>
<tr>
<td>4</td>
<td>defq</td>
<td>slurmhel</td>
<td>fred</td>
<td>PD</td>
<td>0:00</td>
<td></td>
<td>(Resources)</td>
</tr>
<tr>
<td>3556</td>
<td>defq</td>
<td>slurmhel</td>
<td>fred</td>
<td>PD</td>
<td>0:00</td>
<td></td>
<td>(Resources)</td>
</tr>
<tr>
<td>3557</td>
<td>defq</td>
<td>slurmhel</td>
<td>fred</td>
<td>PD</td>
<td>0:00</td>
<td></td>
<td>(Resources)</td>
</tr>
<tr>
<td>3558</td>
<td>defq</td>
<td>slurmhel</td>
<td>fred</td>
<td>PD</td>
<td>0:00</td>
<td></td>
<td>(Resources)</td>
</tr>
<tr>
<td>3559</td>
<td>defq</td>
<td>slurmhel</td>
<td>fred</td>
<td>PD</td>
<td>0:00</td>
<td></td>
<td>(Resources)</td>
</tr>
</tbody>
</table>
The head -4 and tail -4 filters here are convenient for showing just the first 4 rows and last 4 rows of the very long squeue output, and skipping the bulk of the queue.

The preceding output illustrates how, with the jobs queued up, nothing is being processed yet from jobs number 1 to 3559 due to the resources not yet being available.

At this point cm-scale should have noticed that jobs are queued and that resources are needed to handle the jobs.

At the start of this example session the cloud director is not up. So, cm-scale powers it up. This can be seen by running the ds command, or from CMDaemon info messages:

[bright82->device]% ds | grep director
eu-west-1-director [ DOWN ]
then some time later:
eu-west-1-director [ PENDING ] (External ip assigned: 34.249.166.63, setting up tunnel)
then some time later:
eu-west-1-director [ INSTALLING ] (node installer started)
then some time later:
eu-west-1-director [ INSTALLER_CALLINGINIT ] (switching to local root)
then some time later:
eu-west-1-director [ UP ]

If the cloud director is yet to be provisioned to the cloud from the head node for the very first time ("from scratch"), then that can take a while. Then, because the cloud compute nodes are in turn provisioned from the cloud director, it takes a while for the cloud compute nodes to be ready to run the jobs. So, the jobs just have to wait around in the queue until the cloud compute nodes are ready, before they are handled. Fortunately, the startup of a cloud director is by default much faster after the very first time.

A quick aside about how provisioning is speeded up the next time around: The cloud compute nodes will be stopped if they are idle, and after there are no more jobs in the queue, because the jobs have all been dealt with. Then, when the extranodeidletime setting has been exceeded, the cloud director is also stopped. The next time that jobs are queued up, all the cloud nodes are provisioned from a stopped state, rather than from scratch, and so they are ready for job execution much faster. Therefore, unlike the first time, the jobs queued up the next time are processed with less waiting around.

Getting back to how things proceed in the example session after the cloud director is up: cm-scale then provisions the cloud compute nodes eu-west-1-node001 and eu-west-1-node002 from the cloud director.

Example

[bright82->device]% ds | grep cnode
eu-west-1-cnode001 ........ [ PENDING ] (Waiting for instance to start)
eu-west-1-cnode002 ........ [ PENDING ] (Waiting for instance to start)
then some time later:
eu-west-1-cnode002 [ INSTALLING ] (node installer started)
eu-west-1-cnode001 [ INSTALLING ] (node installer started)
and so on

Once these cloud compute nodes reach the state of UP, they can start to process the jobs in the queue. The queue activity then would show something like:

Example

when the dynamic pool nodes are being readied for job execution:

[fred@bright82 ~]$ squeue | head -5 ; squeue | tail -4
JOBID PARTITION  NAME USER ST TIME      NODES NODELIST(REASON)
 1  defq  slurmhel  fred PD 0:00  1 (Resources)
then later:

```
JOBID PARTITION NAME USER ST TIME NODES NODELIST(REASON)
11  defq slurmhel fred CF 0:00 1 eu-west-1-cnode001
12  defq slurmhel fred CF 0:00 1 eu-west-1-cnode002
13  defq slurmhel fred CF 0:00 1 (priority)
14  defq slurmhel fred CF 0:00 1 (priority)
3556 defq slurmhel fred PD 0:00 1 (Priority)
3557 defq slurmhel fred PD 0:00 1 (Priority)
3558 defq slurmhel fred PD 0:00 1 (Priority)
3559 defq slurmhel fred PD 0:00 1 (Priority)
```

then later, when `cm-scale` sees all of the dynamic pool is used up, the lower priority static pool gets started up:

```
JOBID PARTITION NAME USER ST TIME NODES NODELIST(REASON)
165 defq slurmhel fred CF 0:00 1 eu-west-1-cnode001
166 defq slurmhel fred CF 0:00 1 node001
168 defq slurmhel fred CF 0:00 1 node002
3556 defq slurmhel fred PD 0:00 1 (Priority)
3557 defq slurmhel fred PD 0:00 1 (Priority)
3558 defq slurmhel fred PD 0:00 1 (Priority)
3559 defq slurmhel fred PD 0:00 1 (Priority)
167 defq slurmhel fred R 0:00 1 eu-west-1-cnode002
```

In `cmsh`, the priority can be checked with:

Example

```
[bright82->device[bright82]->roles[scaleserver]->resourceproviders]% list
Name (key) Priority Enabled
----------------- ------------ -------
on-premise-nodes 1 yes
pool2 2 yes
```

Also in `cmsh`, the jobs can be listed via `jobs` mode:

Example

```
[bright82->jobs(slurm)]% list | head -5 ; list | tail -4
Type Job ID User Queue Running time Status Nodes
------- ------ ----- ----- ------------ ---------- ------------------
Slurm 334 fred defq 1s COMPLETED eu-west-1-cnode001
Slurm 336 fred defq 1s COMPLETED node001
Slurm 3556 fred defq 0s PENDING
Slurm 3557 fred defq 0s PENDING
Slurm 3558 fred defq 0s PENDING
Slurm 3559 fred defq 0s PENDING
Slurm 335 fred defq 1s RUNNING eu-west-1-cnode002
[bright82->jobs(slurm)]%
```
Eventually, when the queue has been fully processed, the jobs are all gone:

Example

```
fred@bright82 ~]$ squeue
JOBID PARTITION NAME USER ST TIME NODES Nodelist(REASON)

With the current configuration the cloud compute nodes in the dynamic pool pool2 are powered up before the regular compute nodes in the static pool on-premises-pool. That is because the cloud compute nodes have been set by the administrator in this example to have a higher priority. This is typically suboptimal, and is actually configured this way just for illustrative purposes. In a real production cluster, the priority of regular nodes is typically going to be set higher than that for cloud compute nodes, because using on-premise nodes is likely to be cheaper.

The administrator can also check on the job status via the job metrics of cmsh too, using the options to the filter command, such as --pending or --running:

Initially, before the jobs are being run, something like this will show up:

Example

```
[root@bright82 ~]# cmsh -c "jobs ; watch filter --running -u fred"
Every 2.0s: filter --running -u fred Wed Nov 22 16:03:18 2017
No jobs found

Then, eventually, when the jobs are being run, the cloud nodes, which have a higher priority, start job execution, so that the output looks like:

Example

```
Every 2.0s: filter --running -u fred Wed Nov 22 16:50:35 2017
Job ID Job name User Queue Submit time Start time End time Nodes Exit code
------ ------------- ---- ----- ----------- ---------- -------- --------------- ---------
406 slurmhello.sh fred defq 16:16:56 16:27:21 N/A eu-west1-cnode001 0
407 slurmhello.sh fred defq 16:16:56 16:27:21 N/A eu-west1-cnode002 0
```

and eventually the regular on-site nodes which are originally down are started up by the ScaleServer and are also listed.

8.4 Further cm-scale Configuration And Examples

8.4.1 Dynamic Nodes Repurposing

Sometimes it is useful to share the same nodes among several queues, and reuse the nodes for jobs from other queues. This can be done by dynamically assigning node categories in cm-scale. Different settings, or a different software image, then run on the re-assigned node after reprovisioning.

The feature is enabled by setting Assign Category parameter in the tracker settings.

For example, the following case uses Slurm as the workload engine, and sets up two queues chem_q and phys_q. Assuming in this example that jobs that are to go to chem_q require chemistry software on the node, but jobs for phys_q require physics software on the node, and that for some reason the softwares cannot run on the node at the same time. Then, the nodes can be repurposed dynamically. That is, the same node can be used for chemistry or physics jobs by setting up the appropriate configuration for it. In this case the same node can be used by jobs that require a different configuration, software, or even operating system. The trackers configuration may then look as follows:

Example

© Bright Computing, Inc.
Assuming that initially there are two nodes, node001 and node002, both in category chem_cat. Then, when cm-scale finds a pending job in queue phys_q, it may decide to assign category phys_cat to either node001, or to node002. In this way the number of nodes serving queue phys_q increases and number of nodes serving chem_q decreases, in order to handle the current workload. When the job is finished, the old node category is not assigned back to the node, until a new job appears in chem_q and requires this node to have the old category.

8.4.2 Pending Reasons

This section is related only to HPC engines (workload managers). In this section, the term job is used instead of workload.

If cm-scale makes a decision on how many nodes should be started for a job, then it checks the status of the job first. If the job status is pending, then it checks the list of pending reasons for that job. The checks are to find pending reasons that prevent the job from starting when more free nodes become available.

A pending reason can be one of the following 3 types:

**Type 1:** allows a job to start when new free nodes become available

**Type 2:** prevents a job from starting on particular nodes only

**Type 3:** prevents a job from starting anywhere

Each pending reason has a text associated with it. The text is usually printed by the workload manager job statistics utilities. The list of pending reasons texts of types 1 and 2 can be found in the pending reasons exclude file, /cm/local/apps/cm-scale/lib/trackers/hpcqueue/pending_reasons/WLM.exclude, where WLM is a name of workload manager specified in the configuration of the engine in ScaleServer role.

In the pending reasons exclude file, the pending reason texts are listed as one reason per line. The reasons are grouped in two sublists, with headers:

- [IGNORE_ALWAYS]
- [IGNORE_NO_NODE]

© Bright Computing, Inc.
The [IGNORE_ALWAYS] sublist lists the type 1 pending reason texts. If a job has only this group of reasons, then cm-scale considers the job as ready to start, and attempts to create or boot compute nodes for it.

The [IGNORE_NO_NODE] sublist lists the type 2 pending reason texts. If the reason does not specify the hostname of a new free node at the end of a pending reason after the colon (";"), then the job can start on the node. If the reason does specify the hostname of a new free node after the colon, and if the hostname is owned by one of the managed nodes—nodes that can be stopped/started/created by cm-scale—then the job is considered as one that is not to start, when nodes become available.

If a job has a pending reason text that is not in the pending reasons exclude file, then it is assumed to be a type 3 reason. New free nodes for such a job do not get the job started.

If there are several pending reason texts for a job, then cm-scale checks all the pending reasons one by one. If all reasons are from the IGNORE_ALWAYS or IGNORE_NO_NODE sublists, and if a pending reason text matched in the IGNORE_NO_NODE sublist does not include hostnames for the managed nodes, only then will the job be considered as one that can be started just with new nodes.

**Custom Pending Reasons**

If the workload manager supports them, then custom pending reason texts are also supported. The administrator can add a pending reason text to one of the sections in the pending reasons exclude file. The cm-scale service checks only if the pending reason text for the job starts with a text from the pending reasons file. It is therefore enough to specify just a part of the text of the reason in order to make cm-scale take it into account. Regular expressions are also supported. For example, the next two pending reason expressions are equivalent when used to match the pending reason text Not enough job slot(s):

**Example**

- Not enough
- Not enough [a-z]∗ slot(s)

The workload manager statistics utility can be used to find out what custom pending reason texts there are, and to add them to the pending reasons file. To do this, some test job can be forced to have such a pending reason, and the output of the job statistics utility can then be copy-pasted. For example, LSF shows custom pending reasons that look like this:

**Example**

Customized pending reason number <integer>

Here, <integer> is an identifier (an unsigned integer) for the pending reason, as defined by the administrator.

**8.4.3 Locations**

Sometimes it makes sense to restrict the workload manager to run jobs only on a defined subset of nodes. For example, if a user submits a multi-node job, then it is typically better to run all the job processes either on the on-premise nodes, or on the cloud nodes. That is, without mixing the node types used for the job. The locations feature of cm-scale allows this kind of restriction for HPC workload managers.

The cm-scale configuration allows one of these two modes to be selected:

1. **forced location**: when the workload is forced to use one of the locations chosen by cm-scale,
2. **unforced location**: when workloads are free to run on any of the compute nodes that are already managed (running, freed or started) by cm-scale.
In Bright Cluster Manager 8.2, for a forced location, `cm-scale` supports these three different locations:

1. **local**: on-premise nodes,
2. **cloud**: AWS instances (Chapter 3 of the *Cloudbursting Manual*) or Azure instances (Chapter 5 of the *Cloudbursting Manual*)
3. **virtual**: OpenStack Bright-managed virtual nodes (Chapter 5.3 of the *OpenStack Deployment Manual*)

The forced location mode is selected by setting the `Mix Locations` advanced setting in the `scaleserver` role for the node to `yes`. For example, for the head node:

**Example**

```
[bright82->device[bright82]->roles[scaleserver]->advancedsettings]% set mixlocations yes
[bright82->device*[bright82*]->roles*[scaleserver*]->advancedsettings*]% commit
```

For SGE, LSF, and PBS Pro, the compute nodes must already have their configuration defined. These workload managers are therefore not compatible in Bright Cluster Manager 8.2 with the `cm-scale` dynamic resource node provider when using `Mix Locations`, because the nodes could be removed and added automatically, and thus lose the location resource assignment. Bright Cluster Manager 8.2 does not support automatic assignment of the custom resources to the nodes of the mentioned workload managers. For now, only a static resource provider can be used with those workload managers when `Mix Locations` is set to `no`. For Slurm this restriction is not applied.

**Slurm**

Slurm does not allow the assignment of node properties (features, in Slurm terminology) to jobs if no node exists that is labelled by this property. Thus all properties used must be added to some node. This can be the template node in the case of a dynamic resource provider, or it can be an appropriate non-premise node in case of static resource provider use.

For example, in order to assign the `local` location feature to all nodes in the `default` category, the following `cmsh` commands can be executed:

**Example**

```
[root@bright82 ~]# cmsh
[bright82]% category roles default
[bright82->category[default]->roles]% use slurmclient
[bright82->category[default]->roles[slurmclient]]% append features local
[bright82->category*[default*]->roles*[slurmclient*]]% commit
```

**PBS Pro**

A new generic resource, `resources_available.location`, lets the administrator use `cm-scale` to restrict PBS Pro or PBS Pro Community Edition jobs. The resource is added as a string, using the node customization list in the client roles (section 7.11):

**Example**

```
[root@bright82 ~]# cmsh
[bright82]% category roles default
...->roles% use pbsproclient
...->roles[pbsproclient]% nodecustomizations
...->roles[pbsproclient]->nodecustomizations% add resources_available.location
...->roles*[pbsproclient*]->nodecustomizations*[location*]]% set value <location>
...->roles*[pbsproclient*]->nodecustomizations*[location*]]% commit
...->roles[pbsproclient]->nodecustomizations[location]]%
```
Here <location> is one of the three locations supported by cm-scale.

The current location value can be viewed with the qmgr command as follows:

Example

```bash
qmgr -c "print node node001" | grep location
```

**SGE (OGS) and UGE**

A new generic resource, location, lets the administrator use cm-scale to restrict OGS or UGE jobs. The resource is added as a string type complex attribute, using the node customization list in the client roles (section 7.11):

Example

```bash
[root@bright82 ~]# cmsh
[bright82]% category roles default
...->roles] use ugeclient
...->roles[ugeclient]]% nodecustomizations
...->roles[ugeclient]-nodecustomizations]% add location
...->roles[ugeclient]->nodecustomizations*{location=*}% set value <location>
...->roles[ugeclient]->nodecustomizations*{location=*}% commit
...->roles[ugeclient]->nodecustomizations[location]]%
```

LSF

In order to allow cm-scale to restrict LSF jobs, the administrator has to configure a generic resource called location. The resource is added as a dynamic string resource using the role node customization (section 7.11), for instance:

Example

```bash
[root@bright82 ~]# cmsh
[bright82]% category roles default
...->roles] use ugeclient
...->roles[ugeclient]]% nodecustomizations
...->roles[ugeclient]->nodecustomizations]% add location
...->roles[ugeclient]->nodecustomizations*{location=*}% set value <location>
...->roles[ugeclient]->nodecustomizations*{location=*}% commit
...->roles[ugeclient]->nodecustomizations[location]]%
```

Here <location> is one of the three locations supported by cm-scale.

To verify that the resource is added, the following commands can be run:

Example

```bash
[root@bright82 ~]# module load lsf
[root@bright82 ~]# lsinfo | grep location
```

**8.4.4 Azure Storage Accounts Assignment**

If an Azure node is cloned manually from some node or node template, then the Azure node gets the same storage account as the node it has been cloned from. This may slow the nodes down if too many nodes use the same storage account. The cm-scale utility can therefore assign different storage accounts to nodes that are cloned like this.

The maximum number of nodes for such a storage account is defined by the AZURE_DISK_ACCOUNT_NODES parameter. This parameter has a value of 20 by default, and can be
changed in the configuration file /cm/local/apps/cm-scale/lib/config.py. The cm-scale utility must be restarted after the change.

The newly-cloned-by-cm-scale Azure node gets a randomly-generated storage account name if other storage accounts already have enough nodes associated with them. That is, if other storage accounts have AZURE_DISK_ACCOUNT_NODES or more nodes.

The storage account name is assigned in the node cloud settings in storage submode. For example, in cmsh, the assigned storage accounts can be viewed as follows:

Example

```
[root@bright82 ~]# cmsh
[bright82]# device use cnode001
[bright82->device[cnode001]]# cloudsettings
[bright82->device[cnode001]->cloudsettings]# storage
[bright82->...[cnode001]->cloudsettings->storage]# get root-disk storageaccountname
azurepaclogzjus1
[bright82->...[cnode001]->cloudsettings->storage]# get node-installer-disk storageaccountname
azurepaclogzjus1
[bright82->device[cnode001]->cloudsettings->storage]# ..
[bright82->device[cnode001]->cloudsettings]# get bootdiagnosticsstorageaccountname
azurepaclogzjus1
[bright82->device[cnode001]->cloudsettings]# ..
```

If a node is terminated and removed from the Bright configuration, then the storage account remains in Azure. It has to be explicitly manually removed by the administrator.

8.4.5 Mapping HPC Jobs To Particular Nodes

By default, cm-scale assumes that an HPC job submitted to a particular queue can take a node from outside the queue. This is because by assigning a category, or moving the node to a configuration overlay, the node will be moved to the appropriate queue eventually. From this point of view, the nodes form a single resource pool, and the nodes in the pool are repurposed on demand.

In some scenarios there is a need for certain types of HPC jobs run only on particular types of nodes, without the nodes being repurposed. A typical example: jobs with GPU code require cloud nodes that have access to GPU accelerators, while jobs that do not have GPU code can use the less expensive non-GPU cloud nodes. For this case then, the GPU cloud node is started when the GPU job requires a node, and otherwise a non-GPU node is started.

Job segregation is achieved in cm-scale as follows:

1. The Pin Queues setting, which is an advanced setting in the scaleserver role for the node, is enabled:

   ```
   [bright82->device[bright82]->roles[scaleserver]->advancedsettings]# set pinqueues yes
   [bright82->device*[bright82*]->roles*[scaleserver*]->advancedsettings*]# commit
   ```

2. A new queue is created, or an existing one is used. The queue is used for the jobs that require a particular node type.

3. The particular node type is added to this queue. If the node is already defined in Bright Cluster Manager, then the administrator can assign the queue to the node in the workload manager client role. For example, if the workload manager is Slurm, then the queue is assigned to the nodes in the slurmclient role. If the node has not been defined yet and will be cloned on demand (according to the dynamic resource provider settings, page 305), then its template node is assigned to the queue. When a new node is cloned from the template, the queue is then inherited from the template node.

4. The previous two steps are repeated for each job type.
After that, if a user submits a job to one of the queues, then cm-scale starts or clones a node that is linked with the job queue.

The following cmsh session snippet shows a configuration example:

Example

```
[root@bright82 ~]# cmsh
[bright82]# device roles master
[bright82->device[bright82]->roles] use scaleserver
[bright82->...roles[scaleserver]]% resourceproviders
[bright82->...roles[scaleserver]->resourceproviders]% add dynamic rp1
[bright82->...roles[scaleserver]->resourceproviders[rp1]]% set templatenode tnode1
[bright82->...roles[scaleserver]->resourceproviders[rp1]]% set noderange cnode001..cnode100
[bright82->...roles[scaleserver]->resourceproviders[rp1]]% commit
[bright82->...roles[scaleserver]->resourceproviders[rp2]]% clone rp2
[bright82->...roles[scaleserver]->resourceproviders[rp2]]% set templatenode tnode2
[bright82->...roles[scaleserver]->resourceproviders[rp2]]% set noderange cnode101..cnode200
[bright82->...roles[scaleserver]->resourceproviders[rp2]]% commit
[bright82->...roles[scaleserver]]% engines
[bright82->...roles[scaleserver]->engines]% add hpc s1
[bright82->...roles[scaleserver]->engines[e1]]% set workloadmanager slurm
[bright82->...roles[scaleserver]->engines[e1]]% trackers
[bright82->...roles[scaleserver]->engines[e1]->trackers]% add queue tr1
[bright82->...roles[scaleserver]->engines[e1]->trackers[tr1]]% set queue q1
[bright82->...roles[scaleserver]->engines[e1]->trackers[tr1]]% commit
[bright82->...roles[scaleserver]->engines[e1]->trackers[tr2]]% clone tr2
[bright82->...roles[scaleserver]->engines[e1]->trackers[tr2]]% set queue q2
[bright82->...roles[scaleserver]->engines[e1]->trackers[tr2]]% commit
[bright82->...roles[scaleserver]]% category
[bright82->category]% clone default cat1
[bright82->category[cat1]]% roles
[bright82->category[cat1]->roles]% assign slurmclient
[bright82->category[cat1]->roles[slurmclient]]% set queues q1
[bright82->category[cat1]->roles[slurmclient]]% commit
[bright82->category[cat1]]% category clone cat2
[bright82->category[cat2]]% roles
[bright82->category[cat2]->roles[slurmclient]]% set queues q2
[bright82->category[cat2]->roles[slurmclient]]% commit
[bright82->category[cat2]]% device use tnode1
[bright82->device[tnode1]]% set category cat1
[bright82->device[tnode1]]% commit
[bright82->device[tnode1]]% device use tnode2
[bright82->device[tnode2]]% set category cat2
[bright82->device[tnode2]]% commit
[bright82->device[tnode2]]%}
```

Using the preceding configuration, the user may submit a job with a regular workload manager submission utility specifying the queue q1 or q2, depending on whether the job requires nodes that should be cloned from tnode1 or from tnode2.

### 8.4.6 How To Exclude Unused Nodes From Being Stopped

If a node is idle, then by default cm-scale automatically stops or terminates the node.

However, in some cases there may be a need to start a node on demand, and when it becomes idle, there may be a need to keep the node running. This can be useful if the administrator would like to investigate the performance of an application, or to debug some issues. After completing the investigation or debug session, the administrator can stop the node manually.

© Bright Computing, Inc.
The parameter `KEEP_RUNNING_RANGES` keeps such nodes from being stopped or terminated. The parameter should be added to the configuration file `/cm/local/apps/cm-scale/lib/config.py`. To have the changed setting take effect, the `cm-scale` service must be restarted.

`KEEP_RUNNING_RANGES` defines a map of resource provider names to node name ranges. Extra nodes can be added to the range of the nodes. However, if the extra node must not be stopped or terminated by `cm-scale`, then for each resource provider that has such an extra node, the value of `extranodestop` must be set to `yes`.

In the following example, nodes `cnode002`, `cnode003`, `cnode004`, and `cnode010`, are associated with the `azurenodes1` resource provider. They are therefore never stopped or terminated by `cm-scale`. They are only started on demand by `cm-scale`.

The nodes `cnode012` and `cnode014` are associated with the `azurenodes2` resource provider. They are therefore also not stopped or terminated by `cm-scale`.

Example

```python
opts = {
    ...
    "KEEP_RUNNING_RANGES": {
        "azurenodes1": "cnode002..cnode004,cnode010",
        "azurenodes2": "cnode012,cnode014"
    }
}
```

8.4.7 Queue Node Placeholders

A queue node placeholder is a node that does not yet exist, but has a corresponding object that exists, and the object has queues defined, amongst other properties. It can be used to plan resource use.

Job Rejection For Exceeding Total Cluster Resources

At the time of job submission, the workload manager checks the total available number of slots (used and unused) in a queue. This is the sum of the available slots (used and unused) provided by each node in that queue.

- Jobs that require less than the total number of slots are normally made to wait until more slots become available.

- Jobs that require more than this total number of slots are normally rejected outright by the workload manager, without being put into a wait state. This is because workload managers normally follow a logic that relies on the assumption that if the job demands more slots than can exist on the cluster as it is configured at present, then the cluster will never have enough slots to allow a job to run.

Assuming The Resources Can Never Be Provided

The latter assumption, that a cluster will never have enough slots to allow a job to run, is not true when the number of slots is dynamic, as is the case when `cm-scale` is used. When `cm-scale` starts up nodes, it adds them to a job queue, and the workload manager is automatically configured to allow users to submit jobs to the enlarged queue. That is, the newly available slots are configured as soon as possible so that waiting jobs are dealt with as soon as possible. For jobs that have already been rejected, and are not waiting, this is irrelevant, and users would have to submit the jobs once again.

Ideally, in this case, the workload manager should be configured to know about the number of nodes and slots that can be started up in the future, even if they do not exist yet. Based on that, jobs that would normally be rejected, could then also get told to wait until the resources are available, if it turns out that configured future resources will be enough to run the job.

© Bright Computing, Inc.
Slurm Resources Planning With Placeholders:
Slurm allows nodes that do not exist yet to be defined. These are nodes with hostnames that do not resolve, and have the Slurm setting of `state=FUTURE`. Bright Cluster Manager allows Slurm to add such "fake" nodes to Slurm queues dynamically, when not enough real nodes have yet been added. Bright Cluster Manager supports this feature only for Slurm at present.

This feature is not yet implemented for the other workload managers because they require the hostname of nodes that have been added to the workload manager configuration to be resolved.

Within the Slurm server role it is possible to set a list of placeholder objects within the `placeholder` submode of `cmsh`. Each placeholder allows the following values to be set:

- **queue**: the maximum number of nodes
- **maxnodes**: the maximum number of nodes that this queue allows
- **basenodename**: the base node name that is used when a new node name is generated
- **templatename**: a template node that is used to provide user properties taken from its `slurmclient` role when new fake nodes are added.

For example, the following `cmsh` session uses the head node with an existing `slurmserver` role to illustrate how the Slurm queue `defq` could be configured so that it always has a maximum of 32 nodes, with the nodes being like `node001`:

**Example**

```
[root@bright82 ~]# scontrol show part defq | grep " Nodes="
Nodes=node001
[root@bright82 ~]# cmsh
[bright82]% device roles master
[bright82->device[bright82]->roles]% use slurmserver
[bright82->...]->roles[slurmserver]]% placeholders
[bright82->...mserver]->placeholders]% add defq
[bright82->...*]->placeholders*[defq*]]% set maxnodes 32
[bright82->...*]->placeholders*[defq*]]% set basenodename placeholder
[bright82->...*]->placeholders*[defq*]]% set template node001
[bright82->...*]->placeholders*[defq*]]% commit
[bright82->...*]->placeholders*[defq*]]% [root@bright82 ~]# scontrol show part defq | grep " Nodes="
[Nodes=node001,placeholder[01-31]
```

If a new real node is added to the queue, then the number of placeholder nodes is decreased by one.

The placeholders can also be configured in Bright View via the Slurm server role options. For example, for a node node001, using the clickpath:

- Devices→Nodes[node001]→Edit→Settings→Roles→Slurm server role
- →Placeholders
Containerization

Containerization is a technology that allows processes to be isolated by combining cgroups, Linux namespaces, and (container) images.

- Cgroups are described in section 7.10.
- Linux namespaces represent independent spaces for different operating system facilities: process IDs, network interfaces, mount points, inter-process communication resources and others. Such cgroups and namespaces allow processes to be isolated from each other by separating the available resources as much as possible.
- A container image is a component of a container, and is a file that contains one or several layers. The layers cannot be altered as far the container is concerned, and a snapshot of the image can be used for other containers. A union file system is used to combine these layers into a single image. Union file systems allow files and directories of separate file systems to be transparently overlaid, forming a single coherent file system.

Cgroups, namespaces and image are the basis of a container. When the container is created, then a new process can be started within the container. Containerized processes running on a single machine all share the same operating system kernel, so they start instantly. No process is allowed to change the layers of the image. All changes are applied on a temporary layer created on top of the image, and these changes destroyed when the container is removed.

There are several ways to manage the containers, but the most powerful approaches use Docker, also known as Docker Engine (section 9.1), and Kubernetes (section 9.3). Docker manages containers on individual hosts, while Kubernetes manages containers across a cluster. Bright Cluster Manager integrates both of these solutions, so that setup, configuration and monitoring of containers becomes an easily-managed part of Bright Cluster Manager.

9.1 Docker Engine

Docker integration with Bright Cluster Manager 8.2 for Docker version 17.03.2 is available at the time of writing of this paragraph (March 2020) on the x86_64 architecture for RHEL/SL/CentOS versions 7.x, for SLES versions 12 and above, and for Ubuntu 16.04 and above. For a more up-to-date status, the features matrix at https://support.brightcomputing.com/feature-matrix/ can be checked.

Docker Engine (or just Docker) is a tool for container management. Docker allows containers and their images to be created, controlled, and monitored on a host using Docker command line tools or the Docker API.

Swarm mode, which allows containers to spawn on several hosts, is not formally supported by Bright Cluster Manager 8.2. This is because Bright Cluster Manager 8.2 provides Kubernetes for this purpose instead.

Docker provides a utility called docker, and two daemons called dockerd and containerd.

© Bright Computing, Inc.
Additional functionality includes pulling the container image from a specific image registry (section 9.2), configuring the container network, setting systemd limits, and attaching volumes.

### 9.1.1 Docker Setup

Bright Cluster Manager provides the `cm-docker` package. The package includes the following components:

- docker itself, that provides API and delegates the container management to containerd;
- containerd runtime, that manages OCI images and OCI containers (via runC);
- runC, a CLI tool for spawning and running containers according to the OCI specification runtime;
- docker-py, a Python library for the Docker API.

Typically, however, the administrator is expected to simply run the `cm-docker-setup` utility, which is provided by Bright Cluster Manager’s `cm-setup` package. Running `cm-docker-setup` takes care of the installation of the `cm-docker` package and also takes care of Docker setup. If run without options the utility starts up an Ncurses dialog (figure 9.1).

![Figure 9.1: cm-docker-setup Ncurses startup](image)

The `cm-docker-setup` utility asks several questions, such as which Docker registries are to be used, what nodes Docker is to be installed on, which volume back end to configure, and so on. If `cm-docker-setup` is used with the `-c` option, and given a YAML configuration file `<YAMLfile>`, then a runtime configuration is loaded from that file. The YAML file is typically generated and saved from an earlier run.

When the questions in the Ncurses dialog have been answered and the deployment is carried out, the utility:

- installs the `cm-docker` package, if it has not been installed yet
- then assigns the DockerHost role to the node categories or head nodes that were specified
- adds health checks to the Bright Cluster Manager monitoring configuration
- performs the initial configuration of Docker.

The regular nodes on which Docker is to run, are restarted by the utility, if needed. The restart operation provisions the updated images from the image directory onto the nodes.

The `cm-docker` package also includes a module (section 2.2) file, `docker`, which must be loaded in order to use the `docker` command. By default only the administrator can run the `docker` commands after setup (some output ellipsized):

**Example**
9.1 Docker Engine

[9.1.1] example

[root@bright82 ~]# ssh node001
[root@node001 ~]# module load docker
[root@node001 ~]# docker info
Containers: 0
Images: 0
...
Docker Root Dir: /var/lib/docker
[root@node001 ~]#

... Docker Root Dir: /var/lib/docker
[root@node001 ~]#

and the hello-world image can be run as usual with:

Example

[root@node001 ~]# docker run hello-world

Hello from Docker!
This message shows that your installation appears to be working correctly.
...

Or, for example, importing and running Apache containers with Docker may result in the following output:

Example

[root@node001 ~]# module load docker
[root@node001 ~]# docker container ls

CONTAINER ID IMAGE COMMAND CREATED STATUS PORTS NAMES
62071069436f httpd "httpd-foreground" 4 hours ago Up 4 hours 80/tcp elated_euler
3310d85b17f4 httpd "httpd-foreground" 4 hours ago Up 4 hours 80/tcp sleepy_lichterman
35387f3c6606 httpd "httpd-foreground" 4 hours ago Up 4 hours 80/tcp wonderful_wiles
...
[root@node001 ~]#

Using Docker directly means being root on the host. It is rarely sensible to carry out regular user actions as the root user at all times.

So, to make Docker available to regular users, a user management layer and restrictions is provided by Kubernetes.

After Docker has been installed, Kubernetes can be set up to allow regular user access to the Docker containers as covered in section 9.3. It is a best practice for regular users to use Kubernetes instead of Docker commands directly.

9.1.2 Integration With Workload Managers

Bright Cluster Manager does not provide integration of Docker with workload managers. The administrator can however tune the workload managers in some cases to enable Docker support.

- LSF – An open beta version of LSF with Docker support is available from the IBM web site. This LSF version allows jobs to run in Docker containers, and monitors the container resources per job.
- PBS Pro – Altair provides a hook script that allows jobs to start in Docker containers. Altair should be contacted to obtain the script and instructions.

9.1.3 DockerHost Role

When cm-docker-setup is executed, the DockerHost role is assigned to nodes or categories. The DockerHost role is responsible for Docker service management and configuration.

From cmsh, the configuration parameters can be managed from the Docker::Host role:

© Bright Computing, Inc.
Example

[root@bright82 ~]# cmsh
[bright82]# category use default
[bright82->category[default]]% roles
[bright82->category[default]->roles]% assign docker::host
[bright82->category[default]->roles*[Docker::Host]]% show

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add services</td>
<td>yes</td>
</tr>
<tr>
<td>Name</td>
<td>Docker::Host</td>
</tr>
<tr>
<td>Provisioning associations</td>
<td>&lt;0 internally used&gt;</td>
</tr>
<tr>
<td>Revision</td>
<td></td>
</tr>
<tr>
<td>Type</td>
<td>DockerHostRole</td>
</tr>
<tr>
<td>Debug</td>
<td></td>
</tr>
<tr>
<td>Default Ulimits</td>
<td></td>
</tr>
<tr>
<td>Enable SELinux</td>
<td>yes</td>
</tr>
<tr>
<td>Log Level</td>
<td>info</td>
</tr>
<tr>
<td>Spool</td>
<td>/var/lib/docker</td>
</tr>
<tr>
<td>Tmp dir</td>
<td>$spool/tmp</td>
</tr>
<tr>
<td>API Sockets</td>
<td>unix:///var/run/docker.sock</td>
</tr>
<tr>
<td>Bridge</td>
<td></td>
</tr>
<tr>
<td>Bridge IP</td>
<td></td>
</tr>
<tr>
<td>Iptables</td>
<td>yes</td>
</tr>
<tr>
<td>MTU</td>
<td>0</td>
</tr>
<tr>
<td>User Namespace Remap</td>
<td></td>
</tr>
<tr>
<td>Insecure Registries</td>
<td></td>
</tr>
<tr>
<td>Certificates Path</td>
<td>/etc/docker</td>
</tr>
<tr>
<td>Enable TLS</td>
<td>no</td>
</tr>
<tr>
<td>TLS CA</td>
<td></td>
</tr>
<tr>
<td>TLS Certificate</td>
<td></td>
</tr>
<tr>
<td>TLS Key</td>
<td></td>
</tr>
<tr>
<td>Verify TLS</td>
<td>no</td>
</tr>
<tr>
<td>Storage Backends</td>
<td>&lt;0 in submode&gt;</td>
</tr>
<tr>
<td>Containerd Socket</td>
<td></td>
</tr>
<tr>
<td>Runtime</td>
<td>runc</td>
</tr>
<tr>
<td>Options</td>
<td></td>
</tr>
</tbody>
</table>

The parameters that CMDaemon can configure in the DockerHost role, along with a description, are shown in table 9.1:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add services*</td>
<td>Add services to nodes belonging to this node. Care must be taken if setting this to no. (default: yes)</td>
</tr>
<tr>
<td>Debug*</td>
<td>Enable debug mode (default: no)</td>
</tr>
</tbody>
</table>

...continues
### Parameter Description

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Default Ulimits</td>
<td>Set the default ulimit options for all containers</td>
</tr>
<tr>
<td>Enable SELinux*</td>
<td>Enable selinux support in Docker daemon (default: yes)</td>
</tr>
<tr>
<td>Log Level</td>
<td>Set the daemon logging level. In order of increasing verbosity: fatal, error, warn, info, debug. (default: info)</td>
</tr>
<tr>
<td>Spool</td>
<td>Root of the Docker runtime (default: /var/lib/docker)</td>
</tr>
<tr>
<td>Tmp dir</td>
<td>Location for temporary files. Default: $&lt;spool&gt;/tmp, where $&lt;spool&gt; is replaced by the path to the Docker runtime root directory</td>
</tr>
<tr>
<td>API Sockets</td>
<td>Daemon socket(s) to connect to (default: unix:///var/run/docker.socket)</td>
</tr>
<tr>
<td>Bridge</td>
<td>Attach containers to a network bridge (not defined by default)</td>
</tr>
<tr>
<td>Bridge IP</td>
<td>Network bridge IP (not defined by default)</td>
</tr>
<tr>
<td>Iptables*</td>
<td>Enable iptables rules (default: yes)</td>
</tr>
<tr>
<td>MTU</td>
<td>Set the containers network MTU, in bytes (default: 0, which does not set the MTU at all)</td>
</tr>
<tr>
<td>User Namespace Remap</td>
<td>User/Group setting for user namespaces (not defined by default). It can be set to any of &lt;UID&gt;, <a href="">UID:GID</a>, &lt;username&gt;, <a href="">username:groupname</a>. If it is used, then user_namespace.enable=1 must be set in the kernel options for the relevant nodes, and those nodes must be rebooted to pick up the new option.</td>
</tr>
<tr>
<td>Insecure Registries</td>
<td>If registry access uses HTTPS but does not have proper certificates distributed, then the administrator can make Docker accept this situation by adding the registry to this list (empty by default)</td>
</tr>
<tr>
<td>Certificates Path</td>
<td>Path to Docker certificates (default: /etc/docker)</td>
</tr>
<tr>
<td>Enable TLS*</td>
<td>Use TLS (default: no)</td>
</tr>
</tbody>
</table>

© Bright Computing, Inc.
...continued

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>TLS CA</td>
<td>Trust only certificates that are signed by this CA (not defined by default)</td>
</tr>
<tr>
<td>TLS Certificate</td>
<td>Path to TLS certificate file (not defined by default)</td>
</tr>
<tr>
<td>TLS Key</td>
<td>Path to TLS key file (not defined by default)</td>
</tr>
<tr>
<td>Verify TLS*</td>
<td>Use TLS and verify the remote (default: no)</td>
</tr>
<tr>
<td>Storage Backends</td>
<td>Docker storage backends. Storage types can be created and managed, in a</td>
</tr>
<tr>
<td></td>
<td>submode under this mode. The available types are described in table 9.2.</td>
</tr>
<tr>
<td></td>
<td>Each of these storage types has options that can be set from within the</td>
</tr>
<tr>
<td></td>
<td>submode.</td>
</tr>
<tr>
<td>Containerd Socket</td>
<td>Path to containerd socket (default: not used)</td>
</tr>
<tr>
<td>Runtime</td>
<td>Docker runtime</td>
</tr>
<tr>
<td>Options</td>
<td>Additional parameters for docker daemon</td>
</tr>
</tbody>
</table>

* Boolean (takes yes or no as a value)

Table 9.1: DockerHost role and docker options

9.1.4 Iptables

By default iptables rules have been added to nodes that function as a Docker host, to let network traffic go from the containers to outside the pods network. If this conflicts with other software that uses iptables, then this option can be disabled. For example, if the docker::host has already been assigned to the nodes via the default category, then the iptables rules that are set can be disabled by setting the iptables parameter in the Docker::Host role to no:

Example

```
[root@bright82 ~]# cmsh
[bright82]% category use default
[bright82->category[default]]% roles
[bright82->category[default]->roles]% use docker::host
[bright82->category[default]->roles[docker::host]]% set iptables no
[bright82->category[default]->roles*[docker::host]]% commit
```

9.1.5 Storage Back Ends

A core part of the Docker model is the efficient use of containers based on layered images. To implement this Docker provides different storage back ends, also called storage drivers. These storage back ends rely heavily on various filesystem features in the kernel or volume manager. Some storage back ends perform better than others, depending on the circumstances.

The default storage back end configured by cm-docker-setup is overlay2. Storage back ends supported by Docker are listed in table 9.2:

© Bright Computing, Inc.
9.1 Docker Engine

<table>
<thead>
<tr>
<th>Technology</th>
<th>Description</th>
<th>Back end Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>OverlayFS</td>
<td>This is a modern union filesystem. It is the preferred storage driver for recent Docker versions. It has been in the mainline Linux kernel since version 3.18, with additional improvements for Docker in version 4.0. All of the distributions that Bright Cluster Manager 8.2 supports have backported the kernel changes needed for this to work.</td>
<td>overlay2</td>
</tr>
<tr>
<td>Device Mapper</td>
<td>Deprecated in Docker Engine 18.09. It is a kernel-based framework that has been included in the mainline Linux kernel since version 2.6.9. It underpins many advanced volume management technologies on Linux. The driver stores every image and snapshot on its own virtual device, and works at the block level rather than the file level.</td>
<td>devicemapper</td>
</tr>
</tbody>
</table>

- A loopback mechanism can be implemented using loop-lvm mode. This allows files on a local disk to be managed as if they are on a physical disk or block device. This is simpler than the thin pool mode, but is strongly discouraged for production use. In Bright Cluster Manager this mode is implemented by selecting the option loopback (testing only). This is selected in the storage back end selection screen of the cm-docker-setup installation.

- A thin pool mode can be implemented using direct-lvm mode. This uses a logical volume as a thin pool to use as backing for the storage pool, and uses a spare block device. Configuring this is normally more involved. In Bright Cluster Manager this mode is implemented by selecting the option block (production ready). This is selected in the storage back end selection screen of the cm-docker-setup installation session.

Device mapper options for these modes are described in Table 9.3...continues
This was the first storage back end that Docker used. AUFS is not included in the mainline Linux kernel. Out of the distributions that Bright Cluster Manager 8.2 supports, it is only Ubuntu that supports it.

Table 9.2: Docker storage back ends

<table>
<thead>
<tr>
<th>Technology</th>
<th>Description</th>
<th>Back end Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>AUFS</td>
<td>This was the first storage back end that Docker used. AUFS is not included in the mainline Linux kernel. Out of the distributions that Bright Cluster Manager 8.2 supports, it is only Ubuntu that supports it.</td>
<td>aufs</td>
</tr>
</tbody>
</table>

The docker info command, amongst many other items, shows the storage driver and related settings that are being used in Docker:

**Example**

```
[root@bright82 ~]# module load docker
[root@bright82 ~]# docker info
Containers: 29
   Running: 18
   Paused: 0
   Stopped: 11
Images: 3
Server Version: 17.03.2-ce
Storage Driver: devicemapper
   Pool Name: docker-253:17-17199918-pool
   Pool Blocksize: 65.54 kB
   Base Device Size: 10.74 GB
   Backing Filesystem: xfs
   Data file: /dev/loop0
   Metadata file: /dev/loop1
   Data Space Used: 1.296 GB
   Data Space Total: 107.4 GB
   Data Space Available: 4.56 GB
   Metadata Space Used: 4.325 MB
   Metadata Space Total: 2.147 GB
   Metadata Space Available: 2.143 GB
   Thin Pool Minimum Free Space: 10.74 GB
   Udev Sync Supported: true
   Deferred Removal Enabled: false
   Deferred Deletion Enabled: false
   Deferred Deleted Device Count: 0
   Data loop file: /var/lib/docker/devicemapper/data
   Metadata loop file: /var/lib/docker/devicemapper/metadata
   Library Version: 1.02.146-RHEL7 (2018-01-22)
...
```

Docker data volumes are not controlled by the storage driver. Reads and writes to data volumes bypass the storage driver. It is possible to mount any number of data volumes into a container. Multiple containers can also share one or more data volumes.

More information about Docker storage back ends is available at https://docs.docker.com/engine/userguide/storagedriver.

**Device Mapper Driver Settings Support**

Bright Cluster Manager supports device mapper driver settings more explicitly than the other driver back end settings.

By default the device mapper storage back end is added automatically, and can be configured in the storagebackends submode of the DockerHost role:

© Bright Computing, Inc.
Example

```
[bright82->device[bright82]->roles[dockerhost]]% storagebackends
[bright82->device[bright82]->roles[dockerhost]->storagebackends]% use devicemapper
[bright82->device[bright82]->roles[dockerhost]->storagebackends[devicemapper]]% show
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Block Discard</td>
<td>yes</td>
</tr>
<tr>
<td>Block Size</td>
<td>64K</td>
</tr>
<tr>
<td>Filesystem</td>
<td>xfs</td>
</tr>
<tr>
<td>Loop Data Size</td>
<td>100GB</td>
</tr>
<tr>
<td>Loop Device Size</td>
<td>2GB</td>
</tr>
<tr>
<td>Mkfs Arguments Name</td>
<td>devicemapper</td>
</tr>
<tr>
<td>Pool Device</td>
<td>DockerStorageDeviceMapperBackend</td>
</tr>
</tbody>
</table>

The parameters that are used in the Docker device mapper back end are described in table 9.3:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to docker</th>
</tr>
</thead>
<tbody>
<tr>
<td>Blk Discard*</td>
<td>Enables or disables the use of blkdiscard when removing device mapper devices (default: yes)</td>
<td>dm.blkdiscard</td>
</tr>
<tr>
<td>Block Size</td>
<td>Custom blocksize to use for the thin pool (default: 64kB)</td>
<td>dm.blocksize</td>
</tr>
<tr>
<td>Filesystem</td>
<td>Filesystem type to use for the base device (default: xfs)</td>
<td>dm.fs</td>
</tr>
<tr>
<td>Loop Data Size</td>
<td>Size to use when creating the loopback file for the data virtual device which is used for the thin pool (default: 100GB)</td>
<td>dm.loopdatasize</td>
</tr>
<tr>
<td>Loop Device Size</td>
<td>Size to use when creating the base device, which limits the size of images and container (not set by default)</td>
<td>dm.basesize</td>
</tr>
<tr>
<td>Loop Metadata Size</td>
<td>Size to use when creating the loopback file for the metadadata device which is used for the thin pool (default: 2GB)</td>
<td>dm.loopmetadatasize</td>
</tr>
<tr>
<td>Mkfs Arguments</td>
<td>Extra mkfs arguments to be used when creating the base device</td>
<td>dm.mkfsarg</td>
</tr>
<tr>
<td>Mount Options</td>
<td>Extra mount options used when mounting the thin devices</td>
<td>dm.mountopt</td>
</tr>
</tbody>
</table>

...continues

© Bright Computing, Inc.
Parameter | Description | Option to docker
---|---|---
Pool Device | Custom block storage device to use for the thin pool (not set by default) | dm.thinpooldev

* Boolean (takes yes or no as a value)

**Table 9.3: Device mapper back end Docker options**

For back end driver storage settings other than device mapper, such as AUFS or OverlayFS, settings can be added as options if needed. In cmsh this can be done by setting the options parameter in the storagebackend submode under the docker::host role.

### 9.1.6 Docker Monitoring

When cm-docker-setup runs, it configures and runs the following Docker health checks:

1. makes a test API call to the endpoint of the Docker daemon
2. checks containers to see that none is in a dead state

The Docker daemon can be monitored outside of Bright Cluster Manager with the usual commands directly.

Bright Cluster Manager ways to manage or check on Docker include the following:

In CMDaemon, the docker service can be checked:

**Example**

```
[bright82->device[node001]->services]% list
Service (key) Monitored Autostart
------------------------ ---------- ----------
docker yes yes
nslcd yes yes
```

```
[bright82->device[node001]->services]% show docker
Parameter Value
-------------------------------- ------------------------------
Autostart yes
Belongs to role yes
Monitored yes
Revision
Run if ALWAYS
Service docker
Sickness check interval 60
Sickness check script
Sickness check script timeout 10
Timeout -1
```

The docker0 interface statistics can be checked within the nodeoverview output:

**Example**

```
[bright82->device[node001]]% nodeoverview ...

Interface Received Transmitted
------------------------ ------------------------
docker0 16.0 KiB 3.16 KiB
eth0 492 MiB 72.5 MiB
eth1 0 B 0 B
...
The measurable Docker checks if the docker service is healthy.

**Example**

```
[bright82->device[node001]]% dumpmonitoringdata -1h now Docker
```

<table>
<thead>
<tr>
<th>Timestamp</th>
<th>Value</th>
<th>Info</th>
</tr>
</thead>
<tbody>
<tr>
<td>2019/03/19 17:25:17.249</td>
<td>PASS</td>
<td></td>
</tr>
<tr>
<td>2019/03/22 14:59:17.248</td>
<td>PASS</td>
<td></td>
</tr>
</tbody>
</table>

### 9.1.7 Docker Setup For NVIDIA

NVIDIA GPU Cloud (NGC) is a cloud platform that runs on NVIDIA GPUs. NGC containers are lightweight containers that run applications on NVIDIA GPUs. The applications are typically HPC, machine learning, or deep learning applications.

An NGC can be set up to run NGC containers from the registry `http://ngc.nvidia.com`.

Docker can be configured as an NGC running NGC containers by using the NVIDIA Container Runtime for Docker.

The Bright Cluster Manager package provided for this is: `cm-nvidia-docker`. This package can be installed via the package manager, `yum` or `apt`. Dependencies installed include `cm-docker` and `cuda-driver`.

**Example**

```
# yum install cm-nvidia-docker
...
# module load docker
```

The GPU status can then be printed with the NVIDIA system management interface command:

**Example**

```
# docker run --rm nvidia/cuda:9.2-runtime nvidia-smi
Fri Sep 14 11:27:36 2018
+-----------------------------------------------------------------------------+
| NVIDIA-SMI 396.44 Driver Version: 396.44 |                              |
|-------------------------------+----------------------+----------------------+
| GPU Name Persistence-M| Bus-Id Disp.A | Volatile Uncorr. ECC |
| Fan Temp Perf Pwr:Usage/Cap| Memory-Usage | GPU-Util Compute M. | |
|===============================+======================+======================|
| 0 Tesla K40c Off | 00000000:00:08.0 Off | Off |
| 23% 31C P8 23W / 235W | 0MiB / 12206MiB | 0% Default |
+-----------------------------------------------------------------------------+

<table>
<thead>
<tr>
<th>Processes:</th>
<th>GPU Memory</th>
</tr>
</thead>
<tbody>
<tr>
<td>GPU PID Type Process name Usage</td>
<td></td>
</tr>
</tbody>
</table>

| No running processes found |
```

How to run applications in an NGC with Bright Cluster Manager is illustrated for PyTorch, TensorFlow, and MXNet, in the white paper at `https://www.brightcomputing.com/hubfs/docs/Bright-NGC-WPP-v01.pdf`. The sessions illustrated in that paper require the installation of Kubernetes (section 9.3), or Singularity (section 9.4).
9.2 Docker Registries

When a user creates a new container, an image specified by the user should be used. The images are kept either locally on a host, or in a registry. The image registry can be provided by a cloud provider or locally.

9.2.1 Docker And Harbor Registries: Introduction

Docker Hub, A Remote Registry

By default, Docker searches for images in Docker Hub, which is a cloud-hosted public and private image registry. Docker Hub serves a huge collection of existing images that users can make use of. Every user is allowed to create a new account, and to upload and share images with other users. Using the Docker client, a user can search for already-published images, and then pull them down to a host in order to build containers from them.

When an image is found in the registry, then Docker verifies if the latest version of the image has already been downloaded. If it has not, then Docker downloads the images, and stores them locally. It also tries to synchronize them when a new container is created. When the latest image is downloaded, Docker creates a container from the image layers that are formatted to be used by a union file system. Docker can make use of several union file system variants, including AUFS, btrfs, vfs, and DeviceMapper.

Local Image Registry Options: Classic Docker Registry Vs Harbor

Besides using Docker Hub for the image registry, the administrator can also install a local image registry on the cluster. Bright Cluster Manager provides two ways to integrate such a local registry with the cluster, based on existing open source projects:

- The first one is the classic docker registry provided by Docker Inc, and can be useful if the registry is used by trusted users.
- The second registry, Harbor, developed by VMware and introduced in Bright Cluster Manager version 8.1-5, provides additional features such as security and identity management, and is aimed at the enterprise.

Both options can be installed with the `cm-docker-registry-setup` utility, which comes with Bright Cluster Manager’s `cm-setup` package.

9.2.2 Docker And Harbor Registries: Setup And Configuration

Docker Registry and Harbor can be installed via the `cm-docker-registry-setup` command-line utility. They can also be installed via Bright View in Bright Cluster Manager for versions beyond 8.1-6 as follows:

- The Docker Registry Deployment Wizard is launched via the clickpath: `Containers→Docker→Docker Registry Wizard`
- Either Docker Registry, or Harbor, should be chosen as a registry.
- A single node is ticked for the deployment. The address, port, and the root directory for storing the container images are configured. If the head node is selected for Harbor, then the setup later on asks to open the related port on the head node. This is to make Harbor and the Harbor UI externally accessible.
- In the summary page, if the `Ready for deployment` box is ticked, then the administrator can go ahead with deploying the registry.
- When the deployment is complete, the Docker Registry becomes ready for use. Harbor can take a few additional minutes to be ready for use.

© Bright Computing, Inc.
Similar to the case of etcd nodes (section 9.3.2), nodes that run Harbor or Docker Register have the datanode option (section 5.4.4) when installed by Bright Cluster Manager utilities. The option helps prevent the registry being wiped out by accident, and is added when the cm-docker-registry-setup utility is used to install Harbor or Docker Register. This extra protection is put into place because if a registry is wiped out, then the state of images in the container becomes incoherent and cannot be restored.

**Harbor UI** If the head node is where Harbor is to be installed, and is to be made externally accessible, then the Harbor UI can be accessed at https://<head node hostname>:9443.

If a different node is used for Harbor to be installed, then the related port must be forwarded locally.

Harbor can be logged into by default with the admin user and the default Harbor12345 password.

It is recommended to change that password after the first login.

**Clair** Harbor comes with Clair, a tool for vulnerability static analysis of container images.

More information on using Clair with Harbor can be found at https://github.com/vmware/harbor/blob/master/docs/user_guide.md#vulnerability-scanning-via-clair.

**Dealing with a pre-existing Kubernetes or Harbor installation:** Since Harbor uses Docker internally, and because Kubernetes customizes Docker networking, it means that nodes where Kubernetes is running cannot be reused for Harbor, and that nodes where Harbor is running cannot be reused for Kubernetes.

**Docker Registry Daemon Configuration Using The Docker Registry Role**
The Docker Registry role is used to configure and manage the docker-registry daemon, and its parameters are described in table 9.4:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Domain</td>
<td>Main domain name (default: hostname of the node)</td>
</tr>
<tr>
<td>Alt Domains</td>
<td>Alternative domain names (default: FQDN of the node)</td>
</tr>
<tr>
<td>Port</td>
<td>Port (default: 5000)</td>
</tr>
<tr>
<td>Spool Dir</td>
<td>Spool directory (default: /var/lib/docker-registry)</td>
</tr>
</tbody>
</table>

* Boolean (takes yes or no as a value)

**Table 9.4: Docker Registry role parameters**

The values stored in the Docker Registry role are not supposed to be changed, but they are useful for the uninstall procedure, and also as a record of the settings for the administrator.

```
[bright82->device[bright82]->roles[generic::docker_registry]]% environments
[bright82->device[bright82]->roles[generic::docker_registry]->environments]% list
Name (key)    Value       Node Environment
------------------- ------------------- ------------
alt_domains    node001.cm.cluster  no
domain         node001         no
port           5000             no
spool_dir      /var/lib/docker-registry no
```

Further details on the docker-registry daemon can be found at https://github.com/docker/distribution.

© Bright Computing, Inc.
Harbor Daemon Configuration Using The Harbor Role

The Harbor role is used to configure and manage the harbor daemon. The parameters of the role are described in table 9.5:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Domain</td>
<td>Main domain name (default: hostname of the node)</td>
</tr>
<tr>
<td>Alt Domains</td>
<td>Alternative domain names (default: FQDN of the node)</td>
</tr>
<tr>
<td>Port</td>
<td>Port (default: 9443)</td>
</tr>
<tr>
<td>Spool Dir</td>
<td>Spool directory (default: /var/lib/harbor)</td>
</tr>
<tr>
<td>Default Password</td>
<td>Default password of the Harbor admin user (default: Harbor12345)</td>
</tr>
<tr>
<td>DB Password</td>
<td>Password of the Harbor database (default: randomly generated)</td>
</tr>
<tr>
<td>Clair DB Password</td>
<td>Password of the Clair database (default: randomly generated)</td>
</tr>
</tbody>
</table>

* Boolean (takes yes or no as a value)

Table 9.5: Harbor role parameters

The values stored in the Harbor role are not supposed to be changed, but they are useful for the uninstall procedure, and also as reminder of the settings for the administrator.

Further details on Harbor can be found at [https://vmware.github.io/harbor](https://vmware.github.io/harbor).

### 9.3 Kubernetes

Kubernetes is an open-source platform for automating deployment, scaling, and operations of application containers across clusters of hosts. With Kubernetes, it is possible to:

- scale applications on the fly
- seamlessly update running services
- optimize hardware usage by using only the resources that are needed

Bright Cluster Manager provides the administrator with the required packages, allows Kubernetes to be set up on cluster, and manages and monitors Kubernetes. More information about the design
Kubernetes integration with Bright Cluster Manager 8.2 for Kubernetes v1.12.6 is available at the time of writing of this paragraph (March 2020) on the x86_64 architecture for RHEL/SL/CentOS versions 7.x, for SLES versions 12 and above, and for Ubuntu 16.04 and above. For a more up-to-date status, the features matrix at https://support.brightcomputing.com/feature-matrix/ can be checked.

9.3.1 Reference Architecture
A reference architecture for Kubernetes in Bright Cluster Manager comprises:

- **etcd nodes**: An etcd cluster—the Kubernetes distributed key-value storage—runs on regular nodes only, and uses an odd number (1, 3, 5 ...) of nodes.

- **master nodes**: Kubernetes master units run on head or compute nodes. 2 (or 3) are recommended for High Availability (HA). In a Bright HA configuration, both or none of the head nodes should be selected. That is, it must not run on only one head node of an HA configuration.

- **worker nodes**: Kubernetes worker units run on regular nodes only.

Since Bright Cluster Manager version 8.2, multiple clusters of Kubernetes can be deployed. In such a configuration the same node cannot be shared across different Kubernetes clusters.

A Kubernetes API server proxy based on NGINX runs on every node, except for on nodes that run etcd. The proxy also runs on the head node(s).

Because of the server proxy, a port is reserved on the head node(s) for every Kubernetes cluster. This is required for Kubernetes HA (section 9.3.1), and it also allows kubectl and other tools such as Helm to be used from the head node, to access each Kubernetes cluster.

**Kubernetes HA**
For a Kubernetes HA setup the minimum node requirements are:

- at least 3 nodes for Etcd
- at least 2 nodes for Kubernetes Master

In an average cluster, Bright Computing recommends 3 nodes for the etcd cluster, and 3 nodes for the Kubernetes master.

Even without a Kubernetes master in an HA configuration, there is no downtime for existing pods running on the worker nodes. The worker nodes will still continue to work.

However, Kubernetes HA is needed to be able to schedule tasks, spawn new pods, and in general keep the cluster in the desired state.

9.3.2 Kubernetes Setup
Bright Cluster Manager provides the following Kubernetes-related packages:

- The cm-kubernetes-master, cm-kubernetes-node, conntrack and nginx packages: These are always installed on the head node(s) and on the master and worker node(s).

- The cm-etcd package is installed on the nodes selected for etcd. In a similar way to the case of Harbor or Docker Register (section 9.2.2), the nodes that run etcd are protected by Bright Cluster Manager with the datanode option (section 5.4.4). For etcd nodes, the option is added during the cm-kubernetes-setup installation. As in the case for the registries, the datanode option is set in order to help prevent the administrator from wiping out the existing state of etcd nodes. Wiping out the state of etcd nodes means that the Kubernetes cluster becomes incoherent and that it cannot be restored to where it was just before the etcd nodes were wiped.

© Bright Computing, Inc.
The packages are installed automatically from the repository when the administrator runs `cm-kubernetes-setup` from the command line.

The log file produced by the setup can be found in `/var/log/cm-kubernetes-setup.log`.

**Kubernetes Core Addons**

During setup, some critical addon components such as CoreDNS and Calico are automatically deployed in the `kube-system` namespace. In Bright Cluster Manager all addons are treated as Kubernetes applications (section 9.5), and belong to the default app group `system`.

A `cmsh` treeview illustrating the hierarchy to access these applications is:

```
[cmsh]
|-- ...
|-- kubernetes[default]
  |-- appgroups[system]
    |-- applications
```

**CoreDNS:** The DNS server addon for internal service discovery. It reads the IP addresses of services and pods from Etcd, and resolves domain names for them. If a domain name is not found because the domain is external to the Kubernetes cluster, then CoreDNS forwards the request to the main DNS server.

**Calico:** A modern SDN (Software-Defined Networking) addon to manage a cluster-wide network for pods. Calico uses an agent called Felix to run on each node as a pod.

When the Kubernetes cluster is composed of more than 50 nodes, the component Typha is also deployed for better scalability. The number of Typha replicas is calculated by allocating one Typha replica per 150 nodes, with a minimum of 3 (above 50 nodes) and a maximum of 20.

Further details on Calico can be found at https://docs.projectcalico.org/.

**Kubernetes Optional Add-ons**

The following addons are installed by default unless otherwise noted. However, the user can choose to skip some or all of them during the setup.

**Kubernetes Dashboard:** The web user interface addon for GUI cluster administration and metrics visualization.

The Dashboard pod connects to the API server. It can be used


- Users on an external network can log into `kubectl` or Kubernetes Dashboard by following the procedures described in section 9.3.10.

Users on an external network can log into `kubectl` or the Dashboard by following the procedures described in section 9.3.10.

**Heapster:** An addon for Kubernetes cluster container monitoring and performance analysis. The Heapster pod collects and interprets metrics such as compute resource usage, lifecycle events, and so on. It exposes cluster metrics via an API, and enables the HorizontalPodAutoscaler feature.

With Horizontal Pod Autoscaling, Kubernetes automatically scales, based on observed CPU use, the number of pods in a replication controller, a deployment, or replica set. Besides considering the CPU

© Bright Computing, Inc.
usage for scaling, other application-provided metrics may also be used. However, these other metrics are with alpha support only at the time of writing of this section (May 2017).

The Horizontal Pod Autoscaler is implemented as a Kubernetes API resource and a controller. The resource determines the behavior of the controller. The controller periodically adjusts the number of replicas in a replication controller or deployment, in order to match the observed average CPU usage to a target that the user specifies.

**Kubernetes Metrics Server:** This addon is intended as a replacement for Heapster. At the time of writing (March 2019) it is installed by default alongside Heapster, because Heapster is still needed for the Kubernetes Dashboard. The dependency on Heapster is expected to end at some time further in Dashboard development.

**Helm:** An addon that manages charts, which are packages of pre-configured Kubernetes resources. The Helm client and Tiller (Helm server) components are installed and properly configured by default as a Kubernetes addon. They are initialized for every Kubernetes user when the Kubernetes module is loaded.

**NGINX Ingress Controller:** The official Kubernetes ingress controller addon is built around the Kubernetes Ingress resource, using a ConfigMap to store the NGINX configuration. Ingress provides HTTP and HTTPS routes from outside a Kubernetes cluster to services within the cluster. Traffic routing is controlled by rules defined in the Ingress resource.

By default, Bright Cluster Manager provides an ingress for Kubernetes Dashboard during the cm-kubernetes-setup run, so that the Dashboard works. Port 30080 is the default that is set for the HTTP, and port 30443 is the default that is set for HTTPS.

These 2 ports are exposed on every Kubernetes node, both masters and workers.

The Ingress Controller is deployed as a NodePort which means it comes with a default range of possible port values of 30000–32767.

**NVIDIA device plugin for Kubernetes:** An addon option in the cm-kubernetes-setup run. By default it is not selected. To be used, in addition to being selected, it requires that the NVIDIA GPU drivers are first installed (section 7.4 of the Installation Manual) on the head node and regular node container hosts.

This means that, as described in that section, the regular nodes on which the GPUs are located must have the cuda-driver and cuda-dcgm packages installed, inside the software image.

After booting up the container host regular nodes, the functioning of the DCGM (Data Center GPU Manager) tools should be checked by running module load cuda-dcgm; dcgm config --get.

The plugin addon can then be installed as a selection option in the cm-kubernetes-setup session that is run by the system administrator after the NVIDIA GPU drivers have been installed. The plugin is somewhat beta at the time of writing (March 2019).

If cuda-dcgm and cuda-driver are installed for GPUs on the head node, then to ensure that the head node GPUs are detected by CMDaemon, it is recommended to restart CMDaemon on the head node with service cmdaemon restart.

The plugin then allows the GPUs to be consumed from the containers. Overcommitting GPUs (sharing) is not possible from containers or pods. Multiple GPUs can be requested by the container. Some documentation on its use can be found at https://github.com/NVIDIA/k8s-device-plugin.

**Kubernetes Setup From The Command Line**
The cm-kubernetes-setup command line utility has the following usage synopsis:

```
[root@bright82 ~]# cm-kubernetes-setup -h
```
Containerization

usage: Kubernetes Setup cm-kubernetes-setup [-v] [-h] [-c <config_file>]
    [--skip-modules <mod1,mod2,...>]
    [--only-these-modules <mod1,mod2,...>]
    [--dev] [--tests]
    [--undo-on-error]
    [--on-error-action {debug,remotedebug,undo,abort}]
    [--output-remote-execution-runner]
    [--json] [--no-distro-checks]
    [--cluster CLUSTER_NAME]
    [--skip-package-install]
    [--skip-reboot]
    [--skip-image-update]
    [--skip-disksetup-changes]
    [--add-user USERNAME]
    [--namespace NAMESPACE]
    [--role ROLE] [--regenerate-certs]
    [--remove]
    [--yes-i-really-mean-it]
    [--remove-ceph-storage]

optional arguments:
    --cluster CLUSTER_NAME
        Name of the referred Kubernetes cluster

common:
    Common arguments

    -v               Verbose output
    -h, --help       Print this screen
    -c <config_file> Load runtime configuration for plugins from a YAML config file

advanced:
    Various *advanced* configuration options flags.

    --skip-modules <mod1,mod2,...>
        Load and use all the default plugins (kubernetes, docker), except from
        these ones. Comma-separated list.

    --only-these-modules <mod1,mod2,...>
        Out of all default plugins, load and use only those. Comma-separated list.

    --dev
        Enables additional command line arguments

    --tests
        Test integrity of the utility by running Unit Tests

    --undo-on-error
        Upon encountering a critical error, instead of asking the user for choice,
        setup will undo (revert) the deployment stages.

    --on-error-action {debug,remotedebug,undo,abort}
        Upon encountering a critical error, instead of asking the user for choice,
        setup will undo (revert) the deployment stages.

    --output-remote-execution-runner
        Format output for CMDaemon

    --json
        Use json formatting for log lines printed to stdout

    --no-distro-checks
        Disable distribution checks based on ds.json

installing Kubernetes clusters:
    Flags for installing or managing Kubernetes clusters

    --skip-package-install
Skip the package installation steps. Ignores skip_packages flags in the config.

--skip-reboot     Skip the reboot steps.
--skip-image-update Skip the image update steps.
--skip-disksetup-changes

Never change the disk-setup. Use this flag if you manually configure a partition or device for docker thin pool devices for example.

adding user:
Flags for adding a new user in a Kubernetes cluster

--add-user USERNAME  Create a new user in a Kubernetes cluster
--namespace NAMESPACE

Specify namespace for the new (--add-user) role binding (Default: default)
--role ROLE         Specify role for the new (--add-user) role binding (Default: admin)

regenerate Kubernetes certificates:
Flag for regenerating the Kubernetes certificates

--regenerate-certs Regenerate certificates for a Kubernetes cluster

removing Kubernetes clusters:
Flags for removing a Kubernetes cluster

--remove          Remove a Kubernetes cluster
--yes-i-really-mean-it Required for additional safety
--remove-ceph-storage Remove Kubernetes osd pool from Ceph cluster

The cm-kubernetes-setup utility should be executed on the console.

Dealing with a pre-existing Docker installation: Docker is required for Kubernetes configured by Bright Cluster Manager. The setup wizard checks if Docker has been installed (page 336), and automatically installs Docker, if needed. However, if Docker has already been configured on the same category of nodes on which Kubernetes is to be installed, then the installation stops, because overriding the existing Docker configuration may not be what is wanted. To override the existing Docker configuration, Docker for that category should first be removed with the cm-docker-setup --remove command.

Dealing with a pre-existing Etcd cluster: Etcd is required by Kubernetes to store all the key-value states of the Kubernetes cluster. If no Etcd cluster is found, then the setup wizard prompts to deploy an Etcd cluster. If Etcd is already installed, or present from a previous Kubernetes cluster, then the setup wizard prompts on whether to use the existing Etcd cluster.

Dealing with a pre-existing Harbor installation: Since Harbor uses Docker internally, and because Kubernetes customizes Docker networking, it means that nodes where Harbor is running cannot be reused for Kubernetes.

Kubernetes Setup From An Ncurses Session
When the Kubernetes installation is carried out using cm-kubernetes-setup without any options, an Ncurses wizard starts up. The administrator can answer several questions in the wizard. Questions that are asked include questions about which of the node categories or which of the individual nodes should be configured to run the Kubernetes services. There are also questions about the service and pod networks parameters, the port numbers that will be configured for the daemons, whether to install
specific add-ons, and so on. After the wizard has been completed, a configuration file can be saved that can be used to set up Kubernetes.

The configuration file can be deployed immediately from the wizard, or it can be deployed later by specifying it as an option to cm-kubernetes-setup, in the form -c <file>.

Example

[root@bright82 ~]# cm-kubernetes-setup

ncurses session starts up:

![Figure 9.2: Kubernetes Ncurses Session: Main Operations Screen](image)

The deployment via CLI or via Ncurses assigns the appropriate roles, it adds the new Kubernetes cluster, it adds health checks to the monitoring configuration, and it generates certificates for the Kubernetes daemons.

Docker and Etcd are also deployed when needed.

Testing Kubernetes

To test that Kubernetes works, the cluster-info command can be run. For example, on the head node, bright82:

Example

[root@bright82 ~]# kubectl cluster-info
Kubernetes master is running at https://localhost:10443
Heapster is running at https://localhost:10443/api/v1/namespaces/kube-system/services/\heapster/proxy
CoreDNS is running at https://localhost:10443/api/v1/namespaces/kube-system/services/k\ube-dns:dns/proxy

To further debug and diagnose cluster problems, use 'kubectl cluster-info dump'.

After cm-kubernetes-setup finishes, and the regular nodes have been rebooted, the state of the nodes can be found by running the get nodes command:

Example

[root@bright82 ~]# kubectl get nodes
NAME STATUS ROLES AGE VERSION
node001 Ready <none> 25m v1.12.6
node002 Ready <none> 25m v1.12.6
node003 Ready <none> 25m v1.12.6
node004 Ready <none> 25m v1.12.6
node005 Ready <none> 25m v1.12.6
node006 Ready <none> 25m v1.12.6
A six node cluster should show the following Kubernetes installed addons, when using kubectl with the get all -n kube-system option (some lines truncated):

**Example**

```
[root@bright82 ~]# module load kubernetes/default/1.12.6
[root@bright82 ~]# kubectl get all -n kube-system
```

<table>
<thead>
<tr>
<th>NAME</th>
<th>READY</th>
<th>STATUS</th>
<th>RESTARTS</th>
<th>AGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>pod/calico-node-2fh66</td>
<td>2/2</td>
<td>Running</td>
<td>0</td>
<td>32m</td>
</tr>
<tr>
<td>pod/calico-node-5r6p7</td>
<td>2/2</td>
<td>Running</td>
<td>0</td>
<td>32m</td>
</tr>
<tr>
<td>pod/calico-node-886zm</td>
<td>2/2</td>
<td>Running</td>
<td>0</td>
<td>32m</td>
</tr>
<tr>
<td>pod/calico-node-g47b7</td>
<td>2/2</td>
<td>Running</td>
<td>0</td>
<td>32m</td>
</tr>
<tr>
<td>pod/calico-node-ktr5s</td>
<td>2/2</td>
<td>Running</td>
<td>0</td>
<td>32m</td>
</tr>
<tr>
<td>pod/calico-node-lznp7</td>
<td>2/2</td>
<td>Running</td>
<td>0</td>
<td>32m</td>
</tr>
<tr>
<td>pod/coredns-655d564c79-6hhph</td>
<td>1/1</td>
<td>Running</td>
<td>0</td>
<td>32m</td>
</tr>
<tr>
<td>pod/coredns-655d564c79-znhj5</td>
<td>1/1</td>
<td>Running</td>
<td>0</td>
<td>32m</td>
</tr>
<tr>
<td>pod/heapster-85b6d8b5b-k2j9p</td>
<td>1/1</td>
<td>Running</td>
<td>0</td>
<td>32m</td>
</tr>
<tr>
<td>pod/kubernetes-dashboard-77fd78f978-mb2n6</td>
<td>1/1 Running 0 32m</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>pod/metrics-server-5cbcc84f8c-d5kc6</td>
<td>1/1 Running 0 32m</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>pod/metrics-server-5cbcc84f8c-mj88j</td>
<td>1/1 Running 0 32m</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>pod/tiller-deploy-6f6fd74b68-svsfn</td>
<td>1/1 Running 0 32m</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>NAME</th>
<th>TYPE</th>
<th>CLUSTER-IP</th>
<th>EXTERNAL-IP</th>
<th>PORT(S)</th>
</tr>
</thead>
<tbody>
<tr>
<td>service/calico-typha</td>
<td>ClusterIP</td>
<td>10.150.179.56</td>
<td>&lt;none&gt;</td>
<td>5473/TCP</td>
</tr>
<tr>
<td>service/heapster</td>
<td>ClusterIP</td>
<td>10.150.73.227</td>
<td>&lt;none&gt;</td>
<td>80/TCP</td>
</tr>
<tr>
<td>service/kube-dns</td>
<td>ClusterIP</td>
<td>10.150.255.254</td>
<td>&lt;none&gt;</td>
<td>53/UDP,5</td>
</tr>
<tr>
<td>service/kubernetes-dashboard</td>
<td>ClusterIP</td>
<td>10.150.165.90</td>
<td>&lt;none&gt;</td>
<td>443/TCP</td>
</tr>
<tr>
<td>service/tiller-deploy</td>
<td>ClusterIP</td>
<td>10.150.178.166</td>
<td>&lt;none&gt;</td>
<td>44134/TCP</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>NAME</th>
<th>DESIRED</th>
<th>CURRENT</th>
<th>READY</th>
<th>UP-TO-DATE</th>
<th>AVAILABLE</th>
</tr>
</thead>
<tbody>
<tr>
<td>daemonset.apps/calico-node</td>
<td>6</td>
<td>6</td>
<td>6</td>
<td>6</td>
<td>6</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>NAME</th>
<th>DESIRED</th>
<th>CURRENT</th>
<th>UP-TO-DATE</th>
<th>AVAILABLE</th>
</tr>
</thead>
<tbody>
<tr>
<td>deployment.apps/calico-typha</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>deployment.apps/coredns</td>
<td>2</td>
<td>2</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>deployment.apps/heapster</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>deployment.apps/kubernetes-dashboard</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>deployment.apps/metrics-server</td>
<td>2</td>
<td>2</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>deployment.apps/tiller-deploy</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>NAME</th>
<th>DESIRED</th>
<th>CURRENT</th>
<th>READY</th>
<th>AGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>replicaset.apps/calico-typha-5f646c475c</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>32m</td>
</tr>
<tr>
<td>replicaset.apps/coredns-655d564c79</td>
<td>2</td>
<td>2</td>
<td>2</td>
<td>32m</td>
</tr>
<tr>
<td>replicaset.apps/heapster-85b6d8b5b</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>32m</td>
</tr>
<tr>
<td>replicaset.apps/kubernetes-dashboard-77fd78f978</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>32m</td>
</tr>
<tr>
<td>replicaset.apps/metrics-server-5cbcc84f8c</td>
<td>2</td>
<td>2</td>
<td>2</td>
<td>32m</td>
</tr>
<tr>
<td>replicaset.apps/tiller-deploy-6f6fd74b68</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>32m</td>
</tr>
</tbody>
</table>

The administrator can now configure the cluster to suit the particular site requirements.

### 9.3.3 Kubernetes Configuration Overlays

A list of configuration overlays can be seen from within configurationoverlay mode:

**Example**
## 9.3.4 Removing A Kubernetes Cluster

A Kubernetes cluster can be removed using `cm-kubernetes-setup` with the `--remove` and `--yes-i-really-mean-it` options. Also, if there more than one cluster present, then the cluster name must be specified using the `--cluster` parameter.

A removal run looks as follows (some output ellipsized):

### Example

```
[root@bright82 ~]# cm-kubernetes-setup --remove --cluster default --yes-i-really-mean-it
```

Executing 14 stages

```
# Starting execution for 'Kubernetes Setup'
- kubernetes
- docker
# Progress: 0
## stage: kubernetes: Get Kube Cluster
Connecting to CMDaemon
Follow redirection to active head IP: 10.141.255.254
# Progress: 7
## stage: kubernetes: Check Kube Cluster Exists
# Progress: 14
## stage: kubernetes: Find Installed Components
# Progress: 21
## stage: kubernetes: Find Files On Headnodes
# Progress: 28
## stage: kubernetes: Close Api Server Proxy Port
## stage: kubernetes: Remove Installed Components
## stage: kubernetes: Remove Files On Headnodes
## stage: kubernetes: Remove Etcd Spool
## stage: kubernetes: Set Reboot Required
The service nginx on ae-stable-27 was not stopped
You need to reboot 6 nodes to cleanup the network configuration
# Progress: 100
```

Took: 00:04 min.
Progress: 100/100

```
# Finished execution for 'Kubernetes Setup', status: completed
```

Kubernetes Setup finished!

Using the `--remove` option removes the Kubernetes cluster configuration from Bright Cluster Manager, unassigns Kubernetes-related roles—including the EtcdHost role—and removes Kubernetes health
checks. The command does not remove packages that were installed with a `cm-kubernetes-setup` command before that.

After the disabling procedure has finished, the cluster has no Kubernetes configured and running.

### 9.3.5 Kubernetes Cluster Configuration Options

Kubernetes allows many Kubernetes clusters to be configured. These are separated sets of hosts with different certificates, users and other global settings.

When carrying out the Kubernetes setup run, a Kubernetes cluster name will be asked, and a new object with the cluster settings is then added into the CMDaemon configuration. The administrator can change the settings of the cluster from within the `kubernetes` mode of `cmsh` or within the Kubernetes Clusters options window of Bright View, accessible via the clickpath `Containers → Kubernetes Clusters`.

The `cmsh` equivalent looks like:

**Example**

```
[root@bright82 ~]# cmsh
[bright82]% kubernetes list
Name (key)                      
------------------            
default                

[bright82]% kubernetes use default
[bright82->kubernetes[default]]% show

Parameter                          Value
-------------------------------------- --------------------------------------------------------
Notes                                
Revision                             
Name                                 default
Authorization Mode                   Node,RBAC
Kube Config                          /cm/local/apps/kubernetes/var/etc/node.kubeconfig
Kube Client Config                   /cm/local/apps/kubernetes/var/etc/kubelet.kubeconfig
Kube Config Template                 <409 bytes>
CA                                   /cm/local/apps/kubernetes/var/etc/kubeca-default.pem
CA Key                               /cm/local/apps/kubernetes/var/etc/kubeca-default.key
Kubernetes Certificate               /cm/local/apps/kubernetes/var/etc/node.pem
Kubernetes Key                       /cm/local/apps/kubernetes/var/etc/node.key
Kubernetes Client Certificate        /cm/local/apps/kubernetes/var/etc/kubelet.pem
Kubernetes Client Key                /cm/local/apps/kubernetes/var/etc/kubelet.key
Service Accounts Certificate        /cm/local/apps/kubernetes/var/etc/sa-default.pem
Service Accounts Certificate Key    /cm/local/apps/kubernetes/var/etc/sa-default.key
Kubernetes API Aggregator Certificate /cm/local/apps/kubernetes/var/etc/apiaggregator.pem
Kubernetes API Aggregator Certificate Key /cm/local/apps/kubernetes/var/etc/apiaggregator.key
Cluster Domain                       cluster.local
Etcd Cluster                         kube-default
Etcd Prefix                          /kube-apiserver
Etcd Servers                         
Service Network                      kube-default-service
Trusted domains                      bright82,kubernetes,kubernetes.default,kubernetes.default.svc,master,localhost
Pod Network                          kube-default-pod
Pod Network Node Mask                
Internal Network                     internalnet
KubeDNS IP                           10.150.255.254
Kubernetes API server

© Bright Computing, Inc.
Kubernetes API server proxy port 10443
App Groups <1 in submode>
Label Sets <1 in submode>
Module file template <673 bytes>
[bright82->kubernetes[default]]%

The preceding kubernetes mode parameters are described in table 9.6:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Authorization Mode</td>
<td>Selects how to authorize on the secure port (default: RBAC (Role-Based Access Control) and Node Authorization modes)</td>
</tr>
<tr>
<td>Kube Config</td>
<td>Path to a kubeconfig file, specifying how nodes authenticate to the API server</td>
</tr>
<tr>
<td>Kube Client Config</td>
<td>Path to a kubeconfig file, specifying how kubelets authenticate to the API server</td>
</tr>
<tr>
<td>Kube Config Template</td>
<td>Template Bright Cluster Manager uses to generate kubeconfig files for services and users.</td>
</tr>
<tr>
<td>CA</td>
<td>Path to PEM-encoded RSA or ECDSA certificate used for the CA</td>
</tr>
<tr>
<td>CA Key</td>
<td>Path to PEM-encoded RSA or ECDSA private key used for the CA</td>
</tr>
<tr>
<td>Kubernetes Certificate</td>
<td>File containing x509 certificate used by the kubelets</td>
</tr>
<tr>
<td>Kubernetes Key</td>
<td>File containing x509 private key used by the Kubernetes kubelets</td>
</tr>
<tr>
<td>Kubernetes Client Certificate</td>
<td>File containing x509 certificate used for the kubelets</td>
</tr>
<tr>
<td>Kubernetes Client Key</td>
<td>File containing x509 private key used for the Kubernetes kubelets</td>
</tr>
<tr>
<td>Service Accounts Certificate</td>
<td>File containing x509 certificate used for Kubernetes service accounts. This certificate value will be used as --service-account-key-file in the Kubernetes API service.</td>
</tr>
<tr>
<td>Service Accounts Key</td>
<td>File containing x509 private key used for Kubernetes Service Accounts. This key value will be used as --service-account-private-key-file in the controller manager service.</td>
</tr>
<tr>
<td>Cluster Domain</td>
<td>Domain for this cluster</td>
</tr>
<tr>
<td>Etcd Cluster</td>
<td>The Etcd cluster instance.</td>
</tr>
</tbody>
</table>

...continues

© Bright Computing, Inc.
Parameter Description
Etcd Prefix The prefix for all resource paths in etcd
Etcd Servers List of etcd servers to watch (format: http://<IP address>:<port number>)
Service Network Network from which the service cluster IP addresses will be assigned, in IPv4 CIDR format. Must not overlap with any IP address ranges assigned to nodes for pods. Default: 172.29.0.0/16
Pod Network Network where pod IP addresses will be assigned from
Internal Network Network to back the internal communications.
Trusted Domains Trusted domains to be included in Kubernetes-related certificates as Alternative Subject Names.
KubeDNS IP CoreDNS IP Address.
Kubernetes API Server Kubernetes API server address (format: https://<host>:<port number>).
Kubernetes API Server Proxy Port Kubernetes API server proxy port.
Addons Kubernetes Add-ons managed by CMDaemon.

Table 9.6: kubernetes mode parameters

9.3.6 Kubernetes Roles
Kubernetes roles include the following roles:

- EtcdHost (page 361)
- KubernetesApiServerProxy (page 362)
- KubernetesApiServer (page 362)
- KubernetesController (page 365)
- KubernetesScheduler (page 367)
- KubernetesProxy (page 367)
- KubernetesNode (page 368)

When nodes are configured using Kubernetes roles, then settings in these roles may sometimes use the same pointer variables—for example the Kubernetes or Etcd cluster instance. Pointer variables such as these have definitions that are shared across the roles, as indicated by the parameter description tables for the roles, and which are described in the following pages.

© Bright Computing, Inc.
9.3.7 EtcdCluster
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to etcd</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>Etcld Cluster Name.</td>
<td>--initial-cluster-token</td>
</tr>
<tr>
<td>Election Timeout</td>
<td>Election Timeout, in milliseconds.</td>
<td>--election-timeout</td>
</tr>
<tr>
<td>Heart Beat Interval</td>
<td>Heart Beat Interval, in milliseconds.</td>
<td>--heartbeat-interval</td>
</tr>
<tr>
<td>CA</td>
<td>The Certificate Authority (CA) Certificate path for Etcld, used to generate certificates for Etcld.</td>
<td>--peer-trusted-ca-file</td>
</tr>
<tr>
<td>CA Key</td>
<td>The CA Key path for Etcld, used to generate certificates for Etcld.</td>
<td></td>
</tr>
<tr>
<td>Member Certificate</td>
<td>The Certificate path to use for Etcld cluster members, signed with the Etcld CA. The EtcldHost Role can specify a Member CA as well, and in that case it overwrites any value set here.</td>
<td>--peer-cert-file</td>
</tr>
<tr>
<td>Member Certificate</td>
<td>The Key path to use for Etcld cluster members, signed with the Etcld CA.</td>
<td>--peer-key-file</td>
</tr>
<tr>
<td>Key</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Client CA</td>
<td>The CA used for client certificates. When set it is assumed client certificate and key will be generated and signed with this CA by another party. Etcld still expects the path to be correct for the Client Certificate and Key.</td>
<td>--trusted-ca-file</td>
</tr>
<tr>
<td>Client Certificate</td>
<td>The Client Certificate, used by Etcdctl for example.</td>
<td>--cert-file</td>
</tr>
<tr>
<td>Key</td>
<td>The Client Certificate Key, used by Etcdctl for example.</td>
<td>--key-file</td>
</tr>
</tbody>
</table>

* Boolean (takes yes or no as a value)

Table 9.7: EtcldCluster role parameters and etcd options

EtcldHost Role

The EtcldHost role is used to configure and manage the etcd service. The etcd service manages the etcd database, which is a hierarchical distributed key-value database. The database is used by Kubernetes to store its configurations. The EtcldHost role parameters are described in table 9.8:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to etcd</th>
</tr>
</thead>
<tbody>
<tr>
<td>Member Name</td>
<td>The human-readable name for this etcd member ($hostname will be replaced by the node hostname)</td>
<td>--name</td>
</tr>
</tbody>
</table>

...continues

© Bright Computing, Inc.
Containerization

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to <code>etcd</code></th>
</tr>
</thead>
<tbody>
<tr>
<td>Spool</td>
<td>Path to the data directory (default: <code>/var/lib/etcd</code>)</td>
<td>--data-dir</td>
</tr>
<tr>
<td>Advertise Client</td>
<td>List of client URLs for this member to advertise publicly (default: <code>http://$hostname:5001</code>)</td>
<td>--advertise-client-urls</td>
</tr>
<tr>
<td>Advertise Peers</td>
<td>List of peer URLs for this member to advertise to the rest of the cluster (default: <code>http://$hostname:5002</code>)</td>
<td>--initial-advertise-peer-urls</td>
</tr>
<tr>
<td>Listen Peer URLs</td>
<td>List of URLs to listen on for peer traffic (default: <code>http://$hostname:5002</code>)</td>
<td>--listen-peer-urls</td>
</tr>
<tr>
<td>Snapshot Count</td>
<td>Number of committed transactions that trigger a snapshot to disk (default: 5000)</td>
<td>--snapshot-count</td>
</tr>
<tr>
<td>Debug*</td>
<td>Drop the default log level to DEBUG for all subpackages? (default: no)</td>
<td>--debug</td>
</tr>
<tr>
<td>Member Certificate</td>
<td>Etcd member certificate, signed with CA specified in the Etcd Cluster. When set it will overrule the value from the EtcdCluster object. Default empty.</td>
<td>--peer-cert-file</td>
</tr>
<tr>
<td>Member Certificate</td>
<td>Etcd member certificate key, signed with CA specified in the Etcd Cluster. When set it will overrule the value from the EtcdCluster object. Default empty.</td>
<td>--peer-keyt-file</td>
</tr>
</tbody>
</table>

* Boolean (takes yes or no as a value)

Table 9.8: EtcdHost role parameters and `etcd` options

The `etcd` settings are updated by Bright Cluster Manager in `/cm/local/apps/etcd/current/etc/cm-etcd.conf`.

The KubernetesAPIServerProxy Role

The KubernetesAPIServerProxy role sets up a proxy that provides the entrypoint for one or more instances of the Kubernetes API server. The proxy runs on every node of a Kubernetes cluster instance, including the head node.

If multiple Kubernetes master nodes are present, then it enables HA for the Kubernetes master components, as described in section 9.3.1.

The KubernetesAPIServer Role

The KubernetesAPIServer role is used to configure and manage the `kube-apiserver` daemon. The `kube-apiserver` daemon is a Kubernetes API server that validates and configures data for the Kubernetes API objects. The API objects include pods, services, and replication controllers. The API Server
processes REST operations, and provides a front end to the shared state of the cluster through which all the other components interact.

The KubernetesApiServer role parameters are described in table 9.9:
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to <code>kube-apiserver</code></th>
</tr>
</thead>
<tbody>
<tr>
<td>Kubernetes Cluster</td>
<td>The Kubernetes cluster instance (pointer)</td>
<td></td>
</tr>
<tr>
<td>Insecure API Port</td>
<td>The port on which to serve unsecured, unauthenticated access (disabled by default)</td>
<td><code>--insecure-port</code></td>
</tr>
<tr>
<td>Secure API Port</td>
<td>The port on which to serve HTTPS with authentication and authorization. If 0, then HTTPS will not be served at all. (default: 6443)</td>
<td><code>--secure-port</code></td>
</tr>
<tr>
<td>Advertise Address</td>
<td>The IP address on which to advertise the API server to members of the cluster with <code>--advertise-address</code>. If set to 0.0.0.0, then the IP address of the management network of the head node is used. (default: 0.0.0.0)</td>
<td></td>
</tr>
<tr>
<td>Insecure Bind Address</td>
<td>IP address to serve on (default: 127.0.0.1)</td>
<td><code>--insecure-bind-address</code></td>
</tr>
<tr>
<td>Secure Bind Address</td>
<td>The IP address on which to serve the read- and secure ports (default: 0.0.0.0)</td>
<td><code>--bind-address</code></td>
</tr>
<tr>
<td>Admission Control</td>
<td>Ordered list of plug-ins to control the admission of resources into the cluster) (default: NamespaceLifecycle, LimitRanger, ServiceAccount, PersistentVolumeLabel, DefaultStorageClass, ValidatingAdmissionWebhook, ResourceQuota, DefaultTolerationSeconds, MutatingAdmissionWebhook)</td>
<td><code>--admission-control</code></td>
</tr>
<tr>
<td>Allowed Privileged*</td>
<td>If true, allow privileged containers (default: no)</td>
<td><code>--allow-privileged</code></td>
</tr>
<tr>
<td>Event TTL</td>
<td>Time period that events are retained. Empty by default. A format example: 1h0m0s</td>
<td><code>--event-ttl</code></td>
</tr>
<tr>
<td>Kubelet Timeout</td>
<td>Kubelet port timeout (default: 5s)</td>
<td><code>--kubernetes-timeout</code></td>
</tr>
<tr>
<td>Log Level</td>
<td>Log level (default: 0)</td>
<td><code>--v</code></td>
</tr>
<tr>
<td>Log To StdErr*</td>
<td>Logging to stderr means it goes into the systemd journal (default: yes)</td>
<td><code>--logtostderr</code></td>
</tr>
<tr>
<td>Options</td>
<td>Additional parameters for the <code>kube-apiserver</code> daemon (empty by default)</td>
<td></td>
</tr>
</tbody>
</table>

* Boolean (takes `yes` or `no` as a value)

© Bright Computing, Inc.
9.3 Kubernetes

Table 9.9: KubernetesApiServer role parameters and kube-apiserver options

Further details on the Kubernetes API Server can be found at https://kubernetes.io/docs/admin/kube-apiserver/.

KubernetesController Role
The Kubernetes Controller role is used to configure and manage the kube-controller-manager daemon that embeds the core control loops shipped with Kubernetes. In Kubernetes, a controller is a control loop that watches the shared state of the cluster through the API server, and it makes changes in order to try to move the current state towards the desired state. Examples of controllers that ship with Kubernetes at the time of writing (January 2018) are:

- the replication controller
- the endpoints controller,
- the namespace controller,
- the serviceaccounts controller

The KubernetesController role parameters are described in table 9.10:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to</th>
</tr>
</thead>
<tbody>
<tr>
<td>Kubernetes Cluster</td>
<td>The Kubernetes cluster instance (pointer)</td>
<td>kube-controller-manager</td>
</tr>
<tr>
<td>Address</td>
<td>IP address to serve on (default: 0.0.0.0)</td>
<td>--address</td>
</tr>
<tr>
<td>Port</td>
<td>Port to serve on (default: 10252)</td>
<td>--port</td>
</tr>
<tr>
<td>Concurrent Endpoint Syncs</td>
<td>Number of endpoint syncing operations that will be done concurrently. (default: 5)</td>
<td>--concurrent-endpoint-syncs</td>
</tr>
<tr>
<td>Concurrent Rc Syncs</td>
<td>The number of replication controllers that are allowed to sync concurrently. 5</td>
<td>--concurrent-rc-syncs</td>
</tr>
<tr>
<td>Namespace Sync Period</td>
<td>Period for syncing namespace life-cycle updates</td>
<td>--namespace-sync-period</td>
</tr>
<tr>
<td>Node Monitor Grace Period</td>
<td>Period for syncing NodeStatus in NodeController</td>
<td>--node-monitor-grace-period</td>
</tr>
<tr>
<td>Node Monitor Period</td>
<td>Period the running Node is allowed to be unresponsive before marking it unhealthy</td>
<td>--node-monitor-period</td>
</tr>
<tr>
<td>Node Startup Grace Period</td>
<td>Period the starting Node is allowed to be unresponsive before marking it unhealthy</td>
<td>--node-startup-grace-period</td>
</tr>
<tr>
<td>Node Sync Period</td>
<td>Period for syncing nodes from cloud-provider</td>
<td>--node-sync-period</td>
</tr>
</tbody>
</table>

...continues

© Bright Computing, Inc.
...continued

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to kube-controller-manager</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pod Eviction Timeout</td>
<td>Grace period for deleting pods on failed nodes</td>
<td>--pod-eviction-timeout</td>
</tr>
<tr>
<td>Pv Claim Binder Sync Period</td>
<td>Period for syncing persistent volumes and persistent volume claims</td>
<td>--pvclaimbinder-sync-period</td>
</tr>
<tr>
<td>Register Retry Count</td>
<td>Number of retries for initial node registration</td>
<td>--register-retry-count</td>
</tr>
<tr>
<td>Resource Quota Sync Period</td>
<td>Period for syncing quota usage status in the system</td>
<td>--resource-quota-sync-period</td>
</tr>
<tr>
<td>Log Level</td>
<td>Log level (default: 0)</td>
<td>--v</td>
</tr>
<tr>
<td>Log To StdErr*</td>
<td>Logging to stderr means getting it into the systemd journal (default: yes)</td>
<td>--logtostderr</td>
</tr>
<tr>
<td>Options</td>
<td>Additional parameters for the kube-controller-manager daemon</td>
<td></td>
</tr>
<tr>
<td>Cluster signing Cert file</td>
<td>Filename containing a PEM-encoded X509 CA certificate used to issue cluster-scoped certificates. (leave empty to use the value of CA defined in the Kubernetes Cluster instance).</td>
<td>--cluster-signing-cert-file</td>
</tr>
<tr>
<td>Cluster signing Cert Key file</td>
<td>Filename containing a PEM-encoded RSA or ECDSA private key used to sign cluster-scoped certificates. (leave empty to use the value of CA Key defined in the Kubernetes Cluster instance).</td>
<td>--cluster-signing-key-file</td>
</tr>
<tr>
<td>Use Service Account Credentials</td>
<td>Flag to enable or disable use of Service Account Credentials.</td>
<td>--use-service-account-credentials</td>
</tr>
<tr>
<td>Allocate Node Cidrs</td>
<td>Allocate node CIDR in cluster using Pod Network and Node Mask size defined in Kubernetes Cluster Object.</td>
<td>--allocate-node-cidrs</td>
</tr>
<tr>
<td>Kube Config</td>
<td>Path to a kubeconfig file, specifying how to authenticate to API server.</td>
<td>--kubeconfig</td>
</tr>
<tr>
<td>Kubernetes Certificate</td>
<td>File containing x509 Certificate used by Kubernetes Controller Manager. This will be embedded in the Kube Config file.</td>
<td>(1)</td>
</tr>
<tr>
<td>Kubernetes Key</td>
<td>File containing x509 private key used by Kubernetes Controller Manager. This will be embedded in the Kube Config file.</td>
<td>(2)</td>
</tr>
</tbody>
</table>

* Boolean (takes yes or no as a value)

Table 9.10: KubernetesController role parameters and kube-controller-manager options

© Bright Computing, Inc.
Further details on the Kubernetes controller manager can be found at https://kubernetes.io/docs/admin/kube-controller-manager/.

**KubernetesScheduler Role**
The KubernetesScheduler role is used to configure and manage the kube-scheduler daemon. The Kubernetes scheduler defines pod placement, taking into account the individual and collective resource requirements, quality of service requirements, hardware/software/policy constraints, affinity and anti-affinity specifications, data locality, inter-workload interference, deadlines, and so on.

The KubernetesScheduler role parameters are described in table 9.11:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to kube-scheduler</th>
</tr>
</thead>
<tbody>
<tr>
<td>Kubernetes Cluster</td>
<td>The Kubernetes cluster instance (pointer)</td>
<td></td>
</tr>
<tr>
<td>Address</td>
<td>IP address to serve on (default: 0.0.0.0)</td>
<td>--address</td>
</tr>
<tr>
<td>Scheduler Port</td>
<td>Port to serve on (default: 10253)</td>
<td>--port</td>
</tr>
<tr>
<td>Algorithm Provider</td>
<td>The scheduling algorithm provider to use (default: DefaultProvider)</td>
<td>--algorithm-provider</td>
</tr>
<tr>
<td>Policy Config</td>
<td>File with scheduler policy configuration (default: /cm/local/apps/kubernetes/var/etc/scheduler-policy.json)</td>
<td>--policy-config-file</td>
</tr>
<tr>
<td>Log Level</td>
<td>Log level (default: 0)</td>
<td>--v</td>
</tr>
<tr>
<td>Log To StdErr*</td>
<td>Logging to STDERR means getting it into the systemd journal (default: yes)</td>
<td>--logtostderr</td>
</tr>
<tr>
<td>Options</td>
<td>Additional parameters for the kube-scheduler daemon</td>
<td></td>
</tr>
<tr>
<td>Kube Config</td>
<td>Path to a kubeconfig file, specifying how to authenticate to API server.</td>
<td>--kubeconfig</td>
</tr>
<tr>
<td>Kubernetes Certificate</td>
<td>File containing x509 Certificate used by Kubernetes Scheduler. This certificate will be embedded in the Kube Config file.</td>
<td></td>
</tr>
<tr>
<td>Kubernetes Key</td>
<td>File containing x509 private key used by Kubernetes Scheduler. This certificate key will be embedded in the Kube Config file.</td>
<td></td>
</tr>
</tbody>
</table>

* Boolean (takes yes or no as a value)

**Table 9.11: KubernetesScheduler role parameters and kube-scheduler options**

Further details on the Kubernetes scheduler can be found at https://kubernetes.io/docs/admin/kube-scheduler/.

**KubernetesProxy Role**
The KubernetesProxy role is used to configure and manage kube-proxy daemon. The kube-proxy daemon runs on each node, and reflects services as defined in the Kubernetes API. It can do simple TCP and UDP stream-forwarding or round-robin TCP and UDP forwarding across a set of backends.

The KubernetesProxy role parameters are described in table 9.12:
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Configuration Parameter Passed To kube-proxy</th>
</tr>
</thead>
<tbody>
<tr>
<td>Kubernetes Cluster</td>
<td>The Kubernetes cluster instance (pointer)</td>
<td></td>
</tr>
<tr>
<td>Address</td>
<td>IP address to serve on (default: 0.0.0.0)</td>
<td><code>--address</code></td>
</tr>
<tr>
<td>Proxy Port Range Start</td>
<td>Bottom of range of host ports that may be consumed in order to proxy service traffic (not set by default)</td>
<td><code>--proxy-port-range</code></td>
</tr>
<tr>
<td>Proxy Port Range End</td>
<td>Top of range of host ports that may be consumed in order to proxy service traffic (not set by default)</td>
<td><code>--proxy-port-range</code></td>
</tr>
<tr>
<td>Masquerade All</td>
<td>If using the pure iptables proxy, SNAT everything (default: yes)</td>
<td><code>--masquerade-all</code></td>
</tr>
<tr>
<td>Health Check Address</td>
<td>IP address for the health check server to serve on</td>
<td><code>--healthz-port</code></td>
</tr>
<tr>
<td>Health Check Port</td>
<td>Port to bind the health check server to serve on (default: 10251, use 0 to disable)</td>
<td><code>--healthz-port</code></td>
</tr>
<tr>
<td>Oom Score Adj</td>
<td>The oom_score_adj value for the kube-proxy process, in range [-999, 999] (default: -999)</td>
<td><code>--oom-score-adj</code></td>
</tr>
<tr>
<td>Kube Config</td>
<td>Path to a kubeconfig file, specifying how to authenticate to the API server.</td>
<td><code>--kubeconfig</code></td>
</tr>
<tr>
<td>Kubernetes Certificate</td>
<td>File containing x509 Certificate used by kube-proxy. This certificate is embedded in the Kube Config.</td>
<td></td>
</tr>
<tr>
<td>Kubernetes Key</td>
<td>File containing x509 private key used by Kubernetes API server. This certificate key is embedded in the Kube Config.</td>
<td></td>
</tr>
<tr>
<td>Log Level</td>
<td>Log level (default: 0)</td>
<td><code>--v</code></td>
</tr>
<tr>
<td>Log To StdErr*</td>
<td>Logging to STDERR means it goes in the systemd journal (default: yes)</td>
<td><code>--logtostderr</code></td>
</tr>
<tr>
<td>Options</td>
<td>Additional parameters for the kube-scheduler daemon</td>
<td></td>
</tr>
</tbody>
</table>

* Boolean (takes yes or no as a value)

Table 9.12: KubernetesProxy role parameters and kube-proxy options

Further details on the Kubernetes network proxy can be found at https://kubernetes.io/docs/admin/kube-proxy/.

**KubernetesNode Role**

The KubernetesNode role is used to configure and manage the kubelet daemon, which is the primary node agent that runs on each node. The kubelet daemon takes a set of pod specifications, called PodSpecs, and ensures that the containers described in the PodSpecs are running and healthy.

The KubernetesNode role parameters are described in table 9.13:
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to kubelet</th>
</tr>
</thead>
<tbody>
<tr>
<td>Kubernetes Cluster</td>
<td>The Kubernetes cluster instance (pointer)</td>
<td></td>
</tr>
<tr>
<td>Address</td>
<td>IP address to serve on (default: 0.0.0.0)</td>
<td>--address</td>
</tr>
<tr>
<td>Kubelet Port</td>
<td>Port that the HTTP service of the node runs on (default: 10250)</td>
<td>--port</td>
</tr>
<tr>
<td>CNI plugin binaries path</td>
<td>The full path of the directory in which to search for CNI plugin binaries. (default: /cm/local/apps/kubernetes/current/bin/cni)</td>
<td>--cni-bin-dir</td>
</tr>
<tr>
<td>Enable Server*</td>
<td>Enable server mode of Kubelet</td>
<td>--enable-server</td>
</tr>
<tr>
<td>Host Network Sources</td>
<td>List of sources from which Kubelet allows pods use of the host network (default: file)</td>
<td>--host-network-sources</td>
</tr>
<tr>
<td>Hostname Override</td>
<td>If non-empty, use this string as identification instead of the actual hostname (not set by default)</td>
<td>--hostname-override</td>
</tr>
<tr>
<td>Manifests Path</td>
<td>Path to the config file or directory of files (default: /cm/local/apps/kubernetes/var/etc/manifests)</td>
<td>--pod-manifest-path</td>
</tr>
<tr>
<td>Network plugin</td>
<td>The name of the network plugin to be invoked for various events in kubelet/pod lifecycle. (default: cni)</td>
<td>--network-plugin</td>
</tr>
<tr>
<td>Spool</td>
<td>Directory path for managing Kubelet files (default: /cm/local/apps/kubernetes/var/kubelet)</td>
<td>--root-dir</td>
</tr>
<tr>
<td>Cgroup Root</td>
<td>Optional root cgroup to use for pods</td>
<td>--cgroup-root</td>
</tr>
<tr>
<td>Docker Endpoint</td>
<td>Docker endpoint address to connect to (default: unix:///var/run/docker.sock)</td>
<td>--docker-endpoint</td>
</tr>
<tr>
<td>Docker Spool</td>
<td>Absolute path to the Docker state root directory (default: /var/lib/docker)</td>
<td>--docker-root</td>
</tr>
<tr>
<td>Resource Container</td>
<td>Absolute name of the resource-only container to create and run the Kubelet in (default: /kubelet)</td>
<td>--resource-container</td>
</tr>
<tr>
<td>Allowed Privileged*</td>
<td>If true, allow privileged containers (default: no)</td>
<td>--allow-privileged</td>
</tr>
<tr>
<td>Labels</td>
<td>List of node labels</td>
<td></td>
</tr>
<tr>
<td>Register On Start*</td>
<td>Register the node with the API server (default: yes)</td>
<td>--register-node</td>
</tr>
<tr>
<td>Eviction minimum reclaim</td>
<td>Minimum amount of resources reclaimed in an eviction (default: imagefs.available=1Gi)</td>
<td>--eviction-minimum-reclaim</td>
</tr>
</tbody>
</table>

...continues
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to kubelet</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hard eviction</td>
<td>Hard eviction constraints (default: imagefs.available&lt;1%)</td>
<td>--eviction-hard</td>
</tr>
<tr>
<td>Max Pods</td>
<td>Number of pods that can run on this node</td>
<td>--max-pods</td>
</tr>
<tr>
<td>Max pod eviction grace period</td>
<td>Maximum allowed grace period (in seconds) allowed to terminated pods (default: 60)</td>
<td>--eviction-max-pod-grace-period</td>
</tr>
<tr>
<td>Soft eviction</td>
<td>Soft eviction constraints (default: imagefs.available&lt;5%)</td>
<td>--eviction-soft</td>
</tr>
<tr>
<td>Soft eviction grace period</td>
<td>Soft eviction grace period (default: imagefs.available=1m30s)</td>
<td>--eviction-soft-grace-period</td>
</tr>
<tr>
<td>File Check Frequency</td>
<td>Duration between checking configuration files for new data (default: 20s)</td>
<td>--file-check-frequency</td>
</tr>
<tr>
<td>HTTP Flush Frequency</td>
<td>Duration between checking HTTP for new data (default: 20s)</td>
<td>--http-check-frequency</td>
</tr>
<tr>
<td>Node Status Update Frequency</td>
<td>The absolute free disk space, in MB, to maintain (default: 10s)</td>
<td>--node-status-update-frequency</td>
</tr>
<tr>
<td>Run Once*</td>
<td>If true, exit after spawning pods from local manifests or remote URLs (default: no)</td>
<td>--runonce</td>
</tr>
<tr>
<td>Streaming Connection Idle Timeout</td>
<td>Maximum time a streaming connection can be idle before the connection is automatically closed (default: 1h)</td>
<td>--streaming-connection-idle-timeout</td>
</tr>
<tr>
<td>Sync Frequency</td>
<td>Maximum period between synchronizing running containers and config (default: 10s)</td>
<td>--sync-frequency</td>
</tr>
<tr>
<td>Image GC High Threshold</td>
<td>Percent of disk usage after which image garbage collection is always run (default: 90)</td>
<td>--image-gc-high-threshold</td>
</tr>
<tr>
<td>Image GC Low Threshold</td>
<td>Percent of disk usage before which image garbage collection is never run (default: 80)</td>
<td>--image-gc-low-threshold</td>
</tr>
<tr>
<td>Threshold</td>
<td>maintain (default: 256)</td>
<td>threshold-mb</td>
</tr>
<tr>
<td>Oom Score Adjust</td>
<td>The oom_score_adj value for the kube-proxy process, in range [-999, 999] (default: -999)</td>
<td>--oom-score-adj</td>
</tr>
<tr>
<td>Log Level</td>
<td>Log level (default: 0)</td>
<td>--v</td>
</tr>
<tr>
<td>Log To StdErr*</td>
<td>Logging to STDERR means it gets into the systemd journal (default: yes)</td>
<td>--logtostderr</td>
</tr>
<tr>
<td>Options</td>
<td>Additional parameters for the kubescheduler daemon</td>
<td></td>
</tr>
</tbody>
</table>

...continues
Parameter | Description | Option to kubelet
---|---|---
* Boolean (takes yes or no as a value)

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to kubelet</th>
</tr>
</thead>
<tbody>
<tr>
<td>* Boolean (takes yes or no as a value)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 9.13: KubernetesNode role parameters and kubelet options

Further details on the kubelet daemon can be found at https://kubernetes.io/docs/admin/kubelet/.

9.3.8 Security Model

The Kubernetes security model allows authentication using a certificate authority (CA), with the user and daemon certificates signed by a Kubernetes CA. The Kubernetes CA should not be confused with the Bright Cluster Manager CA.

Bright Cluster Manager will create a CA specifically for issuing all Kubernetes-related certificates. The certificates are put into /cm/local/apps/kubernetes/var/etc/ by default, and /etc/kubernetes/ is made a link to this directory.

In Kubernetes terminology a user is a unique identity accessing the Kubernetes API server. The user may be a human or an automated process. For example an admin or a developer are human users, but kubelet represents an infrastructure user. Both types of users are authorized and authenticated in the same way against the API server.

Kubernetes uses client certificates, tokens, or HTTP basic authentication methods to authenticate users for API calls. Bright Cluster Manager configures client certificate usage by default. The authentication is performed by the API server which validates the user certificate using the common name part of the certificate subject.

In Kubernetes, authorization happens as a separate step from authentication. Authorization applies to all HTTP accesses on the main (secure) API server port. Bright Cluster Manager by default enables RBAC (Role-Based Access Control) combined with Node Authorization. The authorization check for any request thus takes the common name and/or organization part of the certificate subject to determine which roles the user or service has associated. Roles carry a certain set of privileges for resources within Kubernetes.

9.3.9 Addition Of New Kubernetes Users And Kubernetes Role Bindings Configuration

New Kubernetes users can be added via the cm-kubernetes-setup Ncurses utility, and choosing the Add user option (figure 9.2). The utility then prompts for the new or existing user, and also prompts for the namespace that the privileges are to be assigned to. Further on, the wizard prompts for the ClusterRole that is to be bound to the user for the given namespace.

Later the RBAC that is generated under the hood for this role binding can be customized within the user mode of cmsh, in the kuberolebinding submode. For example, for the user test:

Example

```
[bright82]% user
[bright82->user]% use test
ok.
[bright82->user[test]]% kuberolebindings
[bright82->user[test]->kuberolebinding]% use default #default is Kube Cluster auto-completed name
[bright82->user[test]->kuberolebinding[default]]% get config
apiVersion: v1
kind: ServiceAccount
metadata:
    name: test
    namespace: default
---
```
apiVersion: rbac.authorization.k8s.io/v1
kind: RoleBinding
metadata:
  name: test
  namespace: default
roleRef:
  apiGroup: rbac.authorization.k8s.io
  kind: ClusterRole
  name: edit
subjects:
- kind: User
  name: test
  namespace: default
- kind: ServiceAccount
  name: test
  namespace: default

The same can be done directly from cmsh.

Example

[bright82] user
[bright82->user] add test
ok.
[bright82->user*[test*]] kuberolebindings
[bright82->user*[test*]->kuberolebinding] add default #auto-completed name of Kube Cluster
ok.
[bright82->user*[test*]->kuberolebinding*[default*]] set config # will open an $EDITOR
...
[bright82->user*[test*]->kuberolebinding*[default*]] commit
Commit user 'test' ... ok.
[bright82->user*[test*]->kuberolebinding*[default*]]%

The command set config opens an editor, and an appropriate RBAC setting for the user can then be copy/pasted. Or it can be modified, if it already exists, which would be the case if the user was added via cm-kubernetes-setup as in the example prior to the preceding example.

How to construct the RBAC from scratch is documented at https://kubernetes.io/docs/admin/authorization/rbac/#kubectl-create-rolebinding.

After committing the user, Bright Cluster Manager generates a certificate and certificate key for the user under the .kube directory within the home directory of the user: $HOME/.kube/admin.pem|key.

It also creates a Kubernetes configuration file for the user at $HOME/.kube/config.

If there is a need to create custom roles, then it is recommended to add a YAML configuration file for the roles in the Kubernetes applications submode.

A cmsh treeview that displays the hierarchy to access the configuration of a Kubernetes application is:

[cmsh]
|-- ...
|-- kubernetes[default]
  |-- appgroups[system]
  |   |-- applications
  |     ' labelsets[ingress_controller]
|-- ...
  ' user

© Bright Computing, Inc.
An application configuration file, such as for the Dashboard application, can be accessed as indicated by the following session:

**Example**

```
[root@bright82 ~]# cmsh
[bright82]# kubernetes use default
[bright82->kubernetes[default]]% appgroups
[bright82->kubernetes[default]->appgroups]% use system
[bright82->kubernetes[default]->appgroups(system)]% show

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>system</td>
</tr>
<tr>
<td>Revision</td>
<td></td>
</tr>
<tr>
<td>Enabled</td>
<td>yes</td>
</tr>
<tr>
<td>applications</td>
<td>&lt;12 in submode&gt;</td>
</tr>
</tbody>
</table>
```

```
[bright82->kubernetes[default]->appgroups(system)->applications]% list

<table>
<thead>
<tr>
<th>Name (key)</th>
<th>Format</th>
<th>Enabled</th>
</tr>
</thead>
<tbody>
<tr>
<td>bootstrap</td>
<td>Yaml</td>
<td>yes</td>
</tr>
<tr>
<td>calico</td>
<td>Yaml</td>
<td>yes</td>
</tr>
<tr>
<td>dashboard</td>
<td>Yaml</td>
<td>yes</td>
</tr>
<tr>
<td>dashboard_ingress</td>
<td>Yaml</td>
<td>yes</td>
</tr>
<tr>
<td>dns</td>
<td>Yaml</td>
<td>yes</td>
</tr>
<tr>
<td>heapster</td>
<td>Yaml</td>
<td>yes</td>
</tr>
<tr>
<td>helm</td>
<td>Yaml</td>
<td>yes</td>
</tr>
<tr>
<td>ingress_controller</td>
<td>Yaml</td>
<td>yes</td>
</tr>
<tr>
<td>kubernetes_ingress</td>
<td>Yaml</td>
<td>no</td>
</tr>
<tr>
<td>metrics_server</td>
<td>Yaml</td>
<td>yes</td>
</tr>
<tr>
<td>nvidia</td>
<td>Yaml</td>
<td>no</td>
</tr>
<tr>
<td>root</td>
<td>Yaml</td>
<td>yes</td>
</tr>
</tbody>
</table>
```

```
[bright82->kubernetes[default]->appgroups(system)->applications][dashboard]% use dashboard
[bright82->kubernetes[default]->appgroups(system)->applications][dashboard]% show

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>dashboard</td>
</tr>
<tr>
<td>Revision</td>
<td></td>
</tr>
<tr>
<td>Format</td>
<td>Yaml</td>
</tr>
<tr>
<td>Enabled</td>
<td>yes</td>
</tr>
<tr>
<td>Config</td>
<td>&lt;3656 bytes&gt;</td>
</tr>
<tr>
<td>Environment</td>
<td>&lt;0 in submode&gt;</td>
</tr>
<tr>
<td>Exclude list</td>
<td></td>
</tr>
</tbody>
</table>
```

---
apiVersion: v1
type: Opaque
```
```
metadata:
  labels:
    k8s-app: kubernetes-dashboard
  name: kubernetes-dashboard
  namespace: kube-system

Bright Cluster Manager then applies the custom YAML files using the Kubernetes API server. The user test should then be able to read pods, while for other resources the user should have no access:

```
[test@bright82 ~]$ module load kubernetes/default/1.12.6
[test@bright82 ~]$ kubectl get pods -n default
NAME           READY STATUS    RESTARTS AGE
---             ------ ------        ------ ----
test-58bd44785c-7f4lm 1/1   Running   0      13m
```

Revoking these rights can be achieved via cmsh as well:

```
[bright82->user[test]->kuberolebinding[default]]% set enabled no
[bright82->user[test]->kuberolebinding[default]]% commit
...  
[test@bright82 ~]$ kubectl get pods -n default
Error from server (Forbidden): pods is forbidden: User "test" cannot list resource "pods" in API group "" in the namespace "default"
```

If there is an error in the YAML configuration for some reason, then an error should show up in cmsh after committing:

```
[bright82->user[test]->kuberolebinding[default]]% commit
Commit user 'test' ...  
Mon Jan 15 15:59:16 2018 [warning] bright82: Error applying role binding with
'kubectl for user: test
For details type: events details 569
[bright82->user[test]->kuberolebinding[default]]
```

### 9.3.10 Providing Access To External Users

To provide access to users on an external network, the requirements are:

- for kubectl, an entry in the company/internal DNS server should resolve the external FQDN to the head node or to one of the nodes where Kubernetes is running;
- for the Kubernetes Dashboard, dashboard is a subdomain that must be included as a DNS entry under the external FQDN.

The external FQDN, which is set during the Kubernetes cluster setup, is the first item in the list of trusted domains. This can be retrieved from the Kubernetes cluster entity with cmsh as follows:

#### Example

```
[bright82->kubernetes[default]]% get trusteddomains
bright82.example.com
kubernetes
kubernetes.default
kubernetes.default.svc
master
localhost
```
In the preceding example, the FQDN of the cluster is bright82.example.com. The cluster administrator managing their own cluster will have another FQDN, and not this FQDN.

For kubectl, the Kubernetes API server proxy port should be open to the external network. The proxy port can be retrieved from the Kubernetes cluster entity as follows:

```
[bright82->kubernetes[default]]% get kubernetesapiserverproxyport
10443
```

For the Kubernetes Dashboard, the Ingress Controller HTTPS port should be open to the external network. This port can be retrieved from the ingress_controller addon environment:

```
Example
```

```
[bright82->kubernetes[default]]% appgroups
[bright82->kubernetes[default]->appgroups]% applications system
[bright82->kubernetes[default]->appgroups->system% applications][ingress_controller]% environment ingress_controller
[bright82->...applications[ingress_controller]->environment]% list
Name (key) Value Nodes environment
--------------------------- --------------------------------------- ------------------
CM_KUBE_EXTERNAL_FQDN     bright82.example.com yes
CM_KUBE_INGRESS_HTTPS_PORT 30443      yes
CM_KUBE_INGRESS_HTTP_PORT  30080      yes
ingress_controller_label   brightcomputing.com/ingress-controller no
replicas                   1             no
```

If exposing the Kubernetes API server to the external network is selected during setup with cm-kubernetes-setup, then the HTTPS and HTTP ports in the preceding example are opened on the Shorewall service that runs on the head node. Exposure to the external network is enabled by default.

**Convention of using a domain name as a prefix label:** in the preceding example, the brightcomputing.com prefix that is part of the value for ingress_controller_label is just a label rather than a domain. The reason that prefix is used is that it simply follows the convention of using domain names as labels, such as is done by the Kubernetes community (domain: kubernetes.io) and RHEL OpenShift (domain: openshift.io). The prefix brightcomputing.com could equally well have been the prefix brightaccess instead. However it is probably less confusing now to follow the established convention. So that is what is done here for the label.

Users can access the Dashboard using dashboard along with the FQDN and the port value, in a URL that has the form:

```
https://dashboard.<CM_KUBE_EXTERNAL_FQDN>:<CM_KUBE_INGRESS_HTTPS_PORT>
```

So, for example, it could be something like:

```
Example
```

```
https://dashboard.bright82.example.com:30443
```

**Ingress configuration for Dashboard in cmsh:** The default Ingress rule described earlier can be found as an object within cmsh:

```
[bright82->kubernetes[default]->appgroups[system]->applications[dashboard_ingress]]% get config
apiVersion: networking.k8s.io/v1beta1
kind: Ingress
metadata:
```
name: kubernetes-dashboard
namespace: kubernetes-dashboard
annotations:
  kubernetes.io/ingress.class: "nginx"
  nginx.ingress.kubernetes.io/secure-backends: "true"
  nginx.ingress.kubernetes.io/ssl-passthrough: "true"
  nginx.ingress.kubernetes.io/backend-protocol: "HTTPS"

spec:
  rules:
  - host: "dashboard.$CM_KUBE_EXTERNAL_FQDN"
    http:
      paths:
      - path: /
        backend:
          serviceName: kubernetes-dashboard
          servicePort: 443

Using `kubectl`, the Ingress resource can be found with:

```
bash$ kubectl get ingress -n kubernetes-dashboard
NAME   HOSTS            ADDRESS          PORTS   AGE
kubernetes-dashboard dashboard.cluster1.local 10.150.153.251 80     45h
```

The official documentation for Ingress, at https://v1-16.docs.kubernetes.io/docs/concepts/services-networking/ingress/, explains it well. Path rewrites without domain names can also be used to set up Ingress with multiple backends (serviceName and servicePort pairs), without having to deal with setting up a DNS.

**Ingress Controller running on compute nodes:** For scenarios where the head node is not involved in a Kubernetes setup, Bright Cluster Manager does not currently set up any forwarding for the Ingress Controller. Bright Cluster Manager does set up an NGINX proxy to expose the Kubernetes API Server in such cases, and accessing the Dashboard can then be done with the `kubectl proxy` approach.

For now a workaround to forward Ingress to a compute node can be achieved with port-forwarding, for example by adding the following line to `/etc/shorewall/rules` in Shorewall:

```
Example

DNAT  net   nat:10.141.0.1:30443 tcp 30443
```

**Using one Ingress controller for multiple Kubernetes clusters:** Bright Cluster Manager does not offer an out-of-the-box solution for one Ingress Controller with multiple Kubernetes clusters. This configuration can be achieved by configuring software such as NGINX to proxy, based on the domain name to the appropriate backend(s).

### 9.3.11 Networking Model

Kubernetes expects all pods to have unique IP addresses, which are reachable from within the cluster. This can be implemented in several ways, including adding pod network interfaces to a network bridge created on each host, or by using 3rd party tools to manage pod virtual networks.

Since Bright Cluster Manager 8.2, the pod network provider is Calico (https://www.projectcalico.org/). Calico uses the Border Gateway Protocol (BGP) to distribute routes for every Kubernetes pod. This allows the the Kubernetes cluster to be integrated without the need for overlays (IP-in-IP). Calico is particularly suitable for large Kubernetes deployments on bare metal, or in private clouds. This is because for larger deployments the performance and complexity costs of overlay networks can become significant.

© Bright Computing, Inc.
9.3.12 Kubernetes Monitoring

When `cm-kubernetes-setup` is run, it configures the following Kubernetes-related health checks:

1. **KubernetesChildNode**: checks if all the expected agents and services are up and running for active nodes
2. **KubernetesComponentsStatus**: checks if all the daemons running on a node are healthy
3. **KubernetesNodesStatus**: checks if Kubernetes nodes have a status of `Ready`
4. **KubernetesPodsStatus**: checks if all the pods are in one of these states: `Running`, `Succeeded`, or `Pending`

9.3.13 Setup Of A Storage Class For Ceph

Pods running on Kubernetes can use Ceph as a distributed storage system to store data in a persistent way.

Instead of creating Kubernetes PersistentVolumes every time, a modern and practical way is using the StorageClass feature.

Further documentation on StorageClass is available at:

- [https://kubernetes.io/docs/concepts/storage/persistent-volumes/#storageclasses](https://kubernetes.io/docs/concepts/storage/persistent-volumes/#storageclasses)

This section assumes a working Ceph cluster. Ceph installation for Bright Cluster Manager is covered in Chapter 4 of the *OpenStack Deployment Manual*.

A new pool `kube` can be created with a replication factor of 3:

**Example**

```bash
[root@bright82 ~]# ceph osd pool create kube 100 100
pool 'kube' created
[root@bright82 ~]# ceph osd pool set kube size 3
set ppool 1 size to 3
[root@bright82 ~]# ceph osd pool set kube min_size 1
set pool 1 min_size to 1
```

The parameters settings in the preceding example are documented at the Ceph website, at

- [http://docs.ceph.com/docs/master/rados/operations/pools/](http://docs.ceph.com/docs/master/rados/operations/pools/) for documentation on Ceph operations
- [http://docs.ceph.com/docs/master/rados/configuration/pool-pg-config-ref/](http://docs.ceph.com/docs/master/rados/configuration/pool-pg-config-ref/) for documentation on Ceph pool and PG (placement group) configuration

The pods of a given namespace have to have access to the Ceph RBD images created to back the volumes.

A *kube* client can be created with:

**Example**

```bash
[root@bright82 ~]# ceph auth get-or-create client.kube mon 'allow r' osd 'allow rwx pool=kube'
[client.kube]
key = AQCnOvdZpYewBBAAWv1d7c7/XbEvj7Q07N0THg==
```

A list of the current users, and their access control can be viewed with (some output elided):

**Example**
9.3 Kubernetes

[root@bright82 ~]# ceph auth list
installed auth entries:

    osd.0
        key: AQD9M/dZw8HPNRAAT+X8mGSGRUKkJ9LnQo38j4EA==
        caps: [mon] allow rwx
caps: [osd] allow *
    osd.1...
    client.admin
        key: AQCnM/dZDNOPMxAAwQ9Y9ADBjV+6i2Uq/ZNqh5A==
        caps: [mds] allow *
caps: [mgr] allow *
caps: [mon] allow *
caps: [osd] allow *
    ...
    client.kube
        key: AQCn0vdZpYewBBAAWv1d7c7/XbEvj7Q07K0THg==
        caps: [mon] allow r
        caps: [osd] allow rwx pool=kube

The admin user must be able to create images in the pool. The admin configuration must therefore look like the section for client.admin in the preceding example.

Similarly, the kube user must be able to map images. The kube configuration must therefore look similar to the section for client.kube in the preceding example.

A Kubernetes secret must be created in the kube-system namespace, using the Ceph admin key:

[root@bright82 ~]# kubectl create secret generic ceph-secret --type="kubernetes.io/rbd" --from-literal=key=$(ceph auth get-key client.admin) --namespace=kube-system
secret "ceph-secret" created

A Kubernetes secret must be created in the default namespace, and in every Kubernetes namespace that needs storage, using the Ceph user key:

[root@bright82 ~]# kubectl create secret generic ceph-secret-user --type="kubernetes.io/rbd" --from-literal=key=$(ceph auth get-key client.kube) --namespace=default
secret "ceph-secret-user" created

Ceph monitor <IP address>:<port> values can be found by running ceph mon stat:

Example

[root@bright82 ~]# ceph mon stat
e1: 3 mons at {node001=10.141.0.1:6789/0,node002=10.141.0.2:6789/0,node003=10.141.0.3:6789/0},
election epoch 38, quorum 0,1,2 node001,node002,node003

A storage-class.yml file can then be created, similar to:

```yaml
apiVersion: storage.k8s.io/v1
kind: StorageClass
metadata:
  name: fast
provisioner: kubernetes.io/rbd
parameters:
  monitors: 10.141.0.1:6789,10.141.0.2:6789,10.141.0.3:6789
  adminId: admin
```

© Bright Computing, Inc.
Containerization

adminSecretName: ceph-secret
adminSecretNamespace: kube-system
pool: kube
userId: kube
userSecretName: ceph-secret-user

Details about the StorageClass parameters can be found at: https://kubernetes.io/docs/concepts/storage/persistent-volumes/#ceph-rbd

The Kubernetes storage class for Ceph RBD can now be created:

[root@bright82 ~]# kubectl apply -f storage-class.yml
storageclass "fast" created

To verify it has been created, the new StorageClass can be listed with:

[root@bright82 ~]# kubectl get sc
NAME      PROVISIONER
fast      kubernetes.io/rbd

9.3.14 Integration With Harbor

In order to spawn pods that use images from the Harbor registry, a secret must first be created with the credentials:

[root@bright82 ~]# kubectl create secret docker-registry myregistrykey
--docker-server=node001:9443 --docker-username=admin --docker-password=Harbor12345

The secret must then be referenced from the pod:

apiVersion: v1
kind: Pod
metadata:
  name: foo
spec:
  containers:
    - name: foo
      image: node001:9443/library/nginx
      imagePullSecrets:
        - name: myregistrykey

Further information on this is available at https://kubernetes.io/docs/concepts/containers/images/#specifying-imagepullsecrets-on-a-pod.

9.4 Singularity

Bright Cluster Manager provides an application containerization tool called Singularity. Singularity is designed to execute containers as if they are just native applications on a host computer, and to work with HPC. Singularity users can therefore run Singularity containers just as they run any other program on an HPC cluster.

9.4.1 Use Cases

Adding Singularity to Bright Cluster Manager brings a stronger integration of containerization with HPC. While Docker and Kubernetes can work within HPC, some drawbacks still prevent the use of HPC resources in the way that HPC users and administrators are used to.

Besides the use of Singularity containers in HPC jobs, Singularity users can create portable images with their applications. Singularity images are files that represent the container filesystem. These images can be copied from one environment (cluster) to another and executed without modification. Thus,
when a user creates a container image file, it is up to the user what files, or which RPMs, to install in the image. For example, the user can create an image file that bundles Open MPI with the user’s application. This guarantees that the application will be able to run if it requires that MPI implementation, even if no MPI libraries are installed on the execution host or if there is some version incompatibility.

There is no need for a special configuration inside workload managers in order to use Singularity. This is because the containers are designed to be run like any application on the system. Users need just to use the image file as the usual script or binary to be executed in their jobscripts or in a shell. The singularity command can also be used to apply special options to the container, when executing the image file in the jobscript or shell.

### 9.4.2 Package cm-singularity

Singularity is packaged for SLES12 and RHEL7 and derivatives. It is available from the YUM or Zypper repositories from version 7.3 of Bright Cluster Manager onwards, and is distributed as a package called cm-singularity. The package should be installed in the software image for each node. The user is able to run a Singularity image only if the Singularity package is installed on the node. In order to allow users to build an image, it makes sense to install the package on the head and login nodes as well. The tool does not provide services that run in the background, so a simple installation of the package is enough to start using it.

Singularity contexts are always run as the user running them. This means that there is no risk in allowing the containers to have access to, and interact with, the file system of the host.

This means that, if an image is created by the root user on a machine, then the files that require root access inside the image, still need to be allowed root permissions on any other machine. Thus, if a user creates an image on a laptop, and adds a file that can be read only by the root user, then when the container is started on another machine by a regular user, that regular user has no access to the root-only readable file inside the container.

While there is no daemon running as root, nor any persistent processes that an attacker may use to escalate privileges, there is a need to run some system calls as root so that the container is encapsulated. For this part of the run flow, there is a single SUID binary called Sexec (Singularity Exec). This is a simple binary that is as small as possible, and which the Singularity developers claim has been audited by multiple security experts.

### 9.4.3 MPI Integration

Because of the nature of Singularity, all MPI implementations should work fine inside a Singularity container. The developers of the tool have spent a lot of effort in making Singularity aware of Open MPI, as well as adding a Singularity module into Open MPI so that running at extreme scale is as efficient as possible. However, in some cases, starting an MPI process may not be as optimal as execution outside the container. So, specifically for Open MPI, Singularity provides a special mechanism to handle the execution of MPI processes. It adds all the MPI processes of the same MPI application to the same container on a host. This also reduces the application startup time. The Open MPI daemon orted in this case is not added to the running container, which means the overhead of starting up daemons is reduced.

When an Open MPI application that has been packaged to an image is started, the following steps take place:

1. `mpirun` is called;
2. `mpirun` forks and executes `orted`;
3. `orted` initializes the PMI (process management interface);
4. `orted` forks as many times as the number of processes per node requested;
5. the container image is started in each fork (because it is the original command specified in `mpirun` arguments);
6. each container process executes the command (that is, the MPI application) passed inside the given container;

7. each of the MPI process links to the dynamic Open MPI library, which loads shared libraries with \texttt{dlopen} system call;

8. Open MPI libraries connect back to the original \texttt{orted} process via PMI;

9. all non-shared memory communication then occurs through the PMI, and then passes on to local network interfaces.

Additional information about Singularity usage can be found in Chapter 11 of the User Manual. The official web site of the tool is https://www.sylabs.io/singularity.

9.5 Kubernetes Apps

Kubernetes addons were introduced in Bright Cluster Manager version 8.1, and could be managed in that version as part of the addons submode of the kubernetes mode in cmsh. In Bright Cluster Manager version 8.2 this feature has been expanded into the Kubernetes Applications & Groups feature. Kubernetes Applications & Groups, less formally called app groups, can be accessed via the appgroups submode of cmsh:

Example

\begin{verbatim}
root@bright82 ~# cmsh
[bright82]% kubernetes
[bright82->kubernetes[default]]% appgroups
[bright82->kubernetes[default]->appgroups]% list
Name (key) Applications
----------------------------------------------
system <13 in submode>
[bright82->kubernetes[default]->appgroups]%
\end{verbatim}

The version 8.1, addons mode parameters are now accessed in version 8.2 via a default system app group instance. The system instance is accessed in the appsgroup submode.

Example

\begin{verbatim}
[bright82->kubernetes[default]->appgroups]% use system
[bright82->kubernetes[default]->appgroups[system]]% show
Parameter Value
----------------------------------------------
Name system
Revision
Enabled yes
applications <13 in submode>
[bright82->kubernetes[default]->appgroups[system]]% applications
Name (key) Format Enabled
------------------------------ ------ -------
bootstrap Yaml yes
calico Yaml yes
dashboard Yaml yes
dashboard_\texttt{ingress} Yaml yes
dns Yaml yes
heapster Yaml yes
helm Yaml yes
\end{verbatim}

© Bright Computing, Inc.
A Kubernetes application can span multiple namespaces. A name in appgroups therefore only exists to group logically-related applications. Each application contains a YAML configuration file, which Bright Cluster Manager synchronizes to the Kubernetes API.

The default system app group is pre-defined. Other app groups can be created as needed. For example, an app group called monitoring could be created to group applications for running Prometheus, node exporters, and anything else related to exposing or viewing Prometheus metrics.

Toggling the Enable parameter of an app group enables or disables all of its application components in Kubernetes. Finer-grained control is possible within the applications mode level, by toggling the enabled parameter per application component instance. For example, within the calico application component instance:

**Example**

```
[bright82->kubernetes[default]->appgroups[system]->applications]% use calico
[bright82->kubernetes[default]->appgroups[system]->applications[calico]]% show
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>calico</td>
</tr>
<tr>
<td>Revision</td>
<td></td>
</tr>
<tr>
<td>Format</td>
<td>Yaml</td>
</tr>
<tr>
<td>Enabled</td>
<td>yes</td>
</tr>
<tr>
<td>Config</td>
<td>&lt;16420 bytes&gt;</td>
</tr>
<tr>
<td>Environment</td>
<td>&lt;3 in submode&gt;</td>
</tr>
<tr>
<td>Exclude list</td>
<td>/etc/cni,/opt/cni,/run/calico,/var/lib/calico</td>
</tr>
</tbody>
</table>

A large YAML configuration file for each application component instance can be configured via the Config parameter property, using the set option of cmsh. This opens up a text editor and allows the environment variables in the YAML configuration file to be managed.

**Exclude list entries** can be set via the Exclude list parameter property to prevent Bright Cluster Manager software image updates from overwriting files or directories important to the given application.

**Environment entries** can be set via the Environment submode. Environment entries are similar to environment variables, and are used to replace variables inside the YAML configuration file. Environment entries can be added to the environment as well, if the Nodes environment value inside the Environment submode is set to yes.

**Example**

```
[bright82->kubernetes[default]->appgroups[system]->applications[calico]]% environment
[bright82->kubernetes[default]->appgroups[system]->applications[calico]->environment]% list
```

<table>
<thead>
<tr>
<th>Name (key)</th>
<th>Value</th>
<th>Nodes environment</th>
</tr>
</thead>
<tbody>
<tr>
<td>calico_typha_replicas</td>
<td>0</td>
<td>no</td>
</tr>
<tr>
<td>calico_typha_service</td>
<td>none</td>
<td>no</td>
</tr>
<tr>
<td>head_node_internal_ip</td>
<td>10.141.255.254</td>
<td>no</td>
</tr>
</tbody>
</table>

In Bright Cluster Manager 8.2, Spark can be installed as a Kubernetes application using cm-spark-setup. This differs from how cm-spark-setup works in version 8.1, where Spark is only installable in a “traditional”, non-containerized way, rather than as a “Spark on Kubernetes” (section 9.6) implementation.

© Bright Computing, Inc.
9.6 Spark On Kubernetes

The Apache Spark project is moving towards running Spark on Kubernetes. This adds a dependency on Kubernetes. On the other hand, there is no longer a need to have Spark Master and Worker services running continuously. Instead, Spark submits jobs directly to the Kubernetes API. Everything needed to execute the Spark workload then spins up on demand inside Kubernetes. Tearing down Kubernetes pods when they are no longer needed is also carried out automatically.

Official documentation for Spark on Kubernetes does a good job of explaining its architecture. For version 2.4.1, which is the Spark release version supported by Bright Cluster Manager 8.2, this documentation is at https://spark.apache.org/docs/2.4.1/running-on-kubernetes.html.

Bright Cluster Manager has been supporting Spark on Kubernetes since version 2.4.0 onwards, and has included support for the following features since that version:

- Support for containerized PySpark and SparkR applications on Kubernetes
- Client mode support that allows users to run interactive applications and notebooks
- Support for mounting certain types of Kubernetes volumes

By default, to run JAR- and/or Python-based jobs, two default containers are provided by Bright Cluster Manager via Docker Hub at:

- https://hub.docker.com/r/brightcomputing/spark/
- https://hub.docker.com/r/brightcomputing/spark-py/

The cluster administrator can also create custom containers by using a provided helper script (/cm/shared/apps/spark/current/bin/docker-image-tool.sh) that comes with Spark on Kubernetes. This is useful when making extra Scala or Python libraries available for the application. The helper script usage is explained in section 10.0.5 of the User Manual.

The containers should be pushed to a Docker registry before they are run inside Kubernetes. The registry can be Docker Hub, or it can be a local docker registry. To deploy a registry with Bright Cluster Manager 8.2, the cm-docker-registry-setup script (section 9.2) can be used.

Section 9.6.3 discusses granting users access to the docker CLI on the cluster, for creating Spark images.

9.6.1 Deploying Spark On Kubernetes

If Kubernetes has been deployed (section 9.3.2), then Spark on Kubernetes can be deployed by running the Bright Cluster Manager cm-spark-setup Ncurses wizard. The wizard is provided by the cm-setup package.

If JupyterHub has already been deployed on the cluster, then a PySpark Kernel is automatically deployed to JupyterHub.

Installing JupyterHub after Spark setup is also fine—the order does not matter. The Spark kernel should become visible in both cases.

The cm-spark-setup setup wizard

If run with the -h option a short help text is displayed:

[root@bright82 ~]# cm-spark-setup -h

Please wait...
       [--tests] [--undo-on-error]
       [--on-error-action {debug,remotedebug,undo,abort}]
       [--output-remote-execution-runner] [--json]
       [--no-distro-checks] [--remove] [--yes]
       [--skip-reboot]

common:
Common arguments

- v                Verbose output
-h, --help        Print this screen
-c <config_file>  Load runtime configuration for plugins from a YAML config file

advanced:
Various *advanced* configuration options flags.

--dev             Enables additional command line arguments
--tests           Test integrity of the utility by running Unit Tests
--undo-on-error   Upon encountering a critical error, instead of asking the user
                  for choice, setup will undo (revert) the deployment stages.
--on-error-action {debug,remotedebug,undo,abort}
                  Upon encountering a critical error, instead of asking the user
                  for choice, setup will undo (revert) the deployment stages.
--output-remote-execution-runner
                  Format output for CMDaemon
--json            Use json formatting for log lines printed to stdout
--no-distro-checks Disable distribution checks based on ds.json

removing spark deployment:
Flags which can be used for removing spark

--remove          Remove every instance of spark
--yes             Force removal without asking for confirmation
--skip-reboot     Don't reboot the nodes

The wizard can be run with a configuration file, using the -c option. Such a configuration file can be generated from an Ncurses-based run.

To run the wizard in Ncurses mode it is run without options. The user is then guided through various installation screens until Spark configuration is completed, and deployment can then begin.

Figure 9.3: cm-spark-setup main operations screen
Example Installation Run

A `cm-spark-setup` deployment session displays something similar to:

**Example**

```
[root@bright82 ~]# cm-spark-setup -c /root/cm-spark-setup.conf
############################### Starting execution for 'Spark Setup'
- spark
  ## Progress: 0
  ### stage: spark: Search Kube Clusters
  Connecting to CMDaemon
  ## Progress: 16
  ### stage: spark: Collection Packages Installer
  ## Progress: 33
  ### stage: spark: spark-auth
  ## Progress: 50
  ### stage: spark: Show Example Usage

The following commands can be used to test drive Spark on k8s:

```bash
module load kubernetes/default
module load spark

spark-submit \
--master k8s://https://localhost:10443 \
--deploy-mode cluster \
--name spark-pi \
--class org.apache.spark.examples.SparkPi \
--conf spark.executor.instances=5 \
--conf spark.kubernetes.container.image=docker.io/brightcomputing/spark:2.4.1 \
--conf spark.kubernetes.authenticate.driver.serviceAccountName=spark \
local:///opt/spark/examples/jars/spark-examples_2.11-2.4.1.jar
```

**Example**

```
## Progress: 66
### stage: spark: Search Jupyter
## Progress: 100

Took: 00:37 min.
Progress: 100/100
############################### Finished execution for 'Spark Setup', status: completed

Spark Setup finished!
```

In the preceding output, Jupyter/JupyterHub is not found and therefore no PySpark kernel is deployed.

**Test Driving Spark On K8s**

Towards the end of the preceding installation output display, an example command is displayed that can be used to test the deployment. A test driver session using that command displays something similar to the following (some output elided):

**Example**

© Bright Computing, Inc.
[root@bright82 ~]# module load kubernetes/default
[root@bright82 ~]# module load spark
[root@bright82 ~]# spark-submit \\
> --master k8s://https://localhost:10443 \\
> --deploy-mode cluster \\
> --name spark-pi \\
> --class org.apache.spark.examples.SparkPi \\
> --conf spark.executor.instances=5 \\
> --conf spark.kubernetes.container.image=docker.io/brightcomputing/spark:2.4.1 \\
> --conf spark.kubernetes.authenticate.driver.serviceAccountName=spark \\
> local:///opt/spark/examples/jars/spark-examples_2.11-2.4.1.jar

pod name: spark-pi-1557138166085-driver
namespace: default
labels: spark-app-selector -> spark-152fe4dea93747908c990c4c97da133f, spark-role -> driver
pod uid: e58702ff-6fe8-11e9-ab66-fa163e53895f
service account name: spark
volumes: spark-local-dir-1, spark-conf-volume, spark-token-xrvvq
node name: N/A
start time: N/A
container images: N/A
phase: Pending
status: []
....
19/05/06 12:24:44 INFO Client: Application spark-pi finished.
19/05/06 12:24:44 INFO ShutdownHookManager: Shutdown hook called
19/05/06 12:24:45 INFO ShutdownHookManager: Deleting directory /tmp/spark-f2676f82-d783-4d4a-abbe-27c00d2350d9

Kubernetes activity can be viewed during the deployment test drive:

Example

[root@bright82 ~]# kubectl get pod
NAMEREADYSTATUSRESTARTSAGE
spark-pi-155714583970-driver1/1Running013s
spark-pi-155714583970-exec-10/1ContainerCreating03s
spark-pi-155714583970-exec-20/1PENDING03s
....

Depending on the state of the submitted job, the output can be quite different.
In the test drive, a driver pod is first created. This in turn spawns five executors for the submitted
job, as specified by the spark-submit command option: --conf spark.executor.instances=5.
The first-time container images still need to be downloaded. The ContainerCreating status may
therefore take a while to change status. After the Spark job runs and finishes, the output might look
similar to:

Example

[root@bright82 ~]# kubectl get pod
NAMEREADYSTATUSRESTARTSAGE
spark-pi-155714583970-exec-50/1Terminating1m1s
spark-pi-155714583970-driver0/1Completed1m33s

The driver pod remains available for a while, so that the job output can be inspected. Logs can be
inspected with a command such as, for example: kubectl logs spark-pi-155714583970-driver.
For longer-running jobs, the Spark UI can be accessed using kubectl port-forward. This is ex-
plained in more detail on page 62 of the User Manual.

© Bright Computing, Inc.
9.6.2 PySpark Jupyter Notebook

Jupyter can be installed as explained in section 4.2 of the Machine Learning Manual.

The cm-spark-setup wizard is then able to detect such an installation. Running it, as root, after Jupyter installation installs the Spark on Kubernetes kernels, and makes them available to users that are Kubernetes users.

For example, if a user joop is made a Kubernetes user (section 9.3.9), then: before cm-spark-setup is run, the Jupyter kernels might show something like:

```
[joop@bright82 ~]$ module load cm-jupyter
[joop@bright82 ~]$ module load cm-jupyterhub
[joop@bright82 ~]$ jupyter kernelspec list
Available kernels:
python2 /cm/shared/apps/jupyter/5.4.1/share/jupyter/kernels/python2
python3 /cm/shared/apps/jupyter/5.4.1/share/jupyter/kernels/python3
```

After running cm-spark-setup as root, the user joop then sees a list of kernels that include the Spark on Kubernetes module:

```
[joop@bright82 ~]$ module load cm-jupyter
[joop@bright82 ~]$ jupyter kernelspec list
Available kernels:
python2 /cm/shared/apps/jupyter/5.4.1/share/jupyter/kernels/python2
python3 /cm/shared/apps/jupyter/5.4.1/share/jupyter/kernels/python3
spark_on_k8s-default_pyspark /home/joop/.local/share/jupyter/kernels/spark_on_k8s-default_pyspark
```

An example notebook can be found in /cm/shared/apps/spark/current/examples, this can be copied into the users home directory to make it visible after logging in via JupyterHub.

Any user wishing to use the Spark kernels must

- have permissions to execute on Kubernetes
- be added via cm-kubernetes-setup
- be known and have a file $HOME/.kube/config

If these requirements are ignored, then JupyterHub will run into issues when the PySpark kernel is run. The thrown Java exceptions that result from these issues are very obscure and difficult to understand.

Adding a user with cmsh: A user test can be added to the cluster with a session such as:

```
[root@bright82 ~]# cmsh
[bright82]% user
[bright82->user]% add test
[bright82->user*[test*]]% set password
enter new password: ******
retype new password: ******
[bright82->user*[test*]]% commit
[bright82->user[test]]% quit
```

Adding a user to Kubernetes: A user test can be added to Kubernetes via a direct CLI operation, using the --add-user option of cm-kubernetes-setup:

```
root@bright82:~# cm-kubernetes-setup --cluster default --add-user test --namespace default \
--role admin
```

Executing 5 stages
9.6 Spark On Kubernetes

########### Starting execution for 'Kubernetes Setup'
- kubernetes
- docker
## Progress: 0
### stage: kubernetes: Get Kube Cluster
Connecting to CMDaemon
## Progress: 20
### stage: kubernetes: Check Extra User
## Progress: 40
### stage: kubernetes: Create Extra User
## Progress: 60
### stage: kubernetes: Init Helm
## Progress: 100

Took: 00:06 min.
Progress: 100/100
########### Finished execution for 'Kubernetes Setup', status: completed

Kubernetes Setup finished!

Connecting to CMDaemon

An Ncurses way to add users and to assign them roles is also possible, by running `cm-kubernetes-setup` without options, and choosing Add user from the menu (figure 9.2).

Kubernetes role binding configuration is discussed in section 9.3.9.

Providing the example Jupyter notebook for the user: An example Jupyter notebook can be provided for the user test, who has a home directory `/home/test`, with a session such as:

```
root@bright82:~# cp /cm/shared/apps/spark/current/examples/pyspark_on_k8s_example.ipynb /home/test/
root@bright82:~# chown test /home/test/pyspark_on_k8s_example.ipynb
```

The user can then login to JupyterHub and run the notebook with the PySpark kernel.

The user can also copy the example themselves, as explained in section 10.0.4 of the User Manual.

![Figure 9.4: The example kernel being executed on JupyterHub](image)

© Bright Computing, Inc.
9.6.3 Providing Custom Docker Images

Allowing users to work with custom Docker images on the cluster requires adding the user to the docker group. This can be carried out with `usermod -aG docker <user>`.

How to use the scripts provided by Apache Spark to build a custom image is described in section 10.0.5 of the User Manual.

9.7 Kubernetes On Edge

How edge sites can be configured is described in Chapter 2 of the Edge Manual.

If there are Bright Edge sites configured in the cluster, then the Kubernetes setup prompts the user with edge sites that Kubernetes can be deployed on.

![cm-kubernetes-setup prompting for edge sites.](image)

Figure 9.5: cm-kubernetes-setup prompting for edge sites.

If an edge site is selected, then the rest of the wizard prompts only for nodes available within that edge site, prompts only for the associated network interfaces, and so on.

9.7.1 Flags For Edge Installation

Edge directors often lack high-bandwidth connectivity to the central head node, or they often may benefit from coming up as quickly as possible. It can therefore sometimes be useful to skip stages of the setup.

Running `cm-kubernetes-setup --help` displays some additional flags that allow some setup stages, that bring up a cloud director, to be skipped explicitly:

```
cm-kubernetes-setup --help
...  
installing Kubernetes clusters:
  Flags for installing or managing Kubernetes clusters
    --skip-package-install  
      Skip the package installation steps. Ignores skip_packages flags in the config.
    --skip-reboot
      Skip the reboot steps.
    --skip-image-update
      Skip the image update steps.
    --skip-disksetup-changes
      Never change the disk-setup. Use this flag if you manually configure a partition or device for docker thin pool devices for example.
...
```

**Speeding Up Kubernetes Installation To Edge Nodes With The --skip-* Flags: Use Cases**

Explanations and use cases for these flags are given in the following table:
<table>
<thead>
<tr>
<th>Flag</th>
<th>Use case</th>
</tr>
</thead>
<tbody>
<tr>
<td>--skip-package-install</td>
<td>all edge directors share the same software image, and the image is already up to date. So the installer does not need to install packages from that image to the edge director.</td>
</tr>
<tr>
<td>--skip-image-update and --skip-reboot</td>
<td>all edge directors are already provisioned with the up-to-date software image. So the installer does not need to carry out an update from the ISO or head node, and then reboot the edge director.</td>
</tr>
<tr>
<td>--skip-disksetup-changes</td>
<td>all edge directors already have the correct disk layout. This flag can be set if the disk layout was already configured up-front, in order to avoid full provisioning.</td>
</tr>
</tbody>
</table>

These flags can also be configured in the YAML configuration file of the `cm-kubernetes-setup` wizard.

The flags can be used for scripted installations for quick Kubernetes setups. For a scripted installation of an edge director, preparations can be done beforehand so that all the requirements in the software images that the edge directors use are already installed, the right disk layouts are already configured, and packages are already updated.

All the stages in the flag options can then be skipped for installing onto edge sites. This can make the setup take just a few seconds per Kubernetes deployment.

### 9.8 Kubernetes On AWS

By default, EC2 does source/destination checking on instances running within it. The source/destination check allows only traffic that has the instance as its source or destination.

Kubernetes on a Bright-managed cluster in AWS would normally run into the source/destination check. For example, if there are Kubernetes pods on two different nodes, then when they communicate with each other, using the pod network IP addresses, the pod IP addresses differ from the actual internal IP address of the node instance inside AWS. For example, a pod IP address may be, say, 172.29.0.2, while a node IP address might be of the dotted quad form 10.x.y.z.

The source/destination check setting must be disabled on each instance that is part of the Kubernetes cluster. Disabling the check can be done via:

- the AWS management console (figure 9.6)
- the API, by running `cm-kubernetes-setup --fix-nat <cloud provider instance>` on the head node.

By default, `<cloud provider instance>` takes the value `amazon` when a cloud provider object is created by Bright Cluster Manager by the cluster administrator. The cloud provider names can be seen within the cloud mode of `cmsh`. 

© Bright Computing, Inc.
9.9 Kubernetes On Azure

Kubernetes on a Bright-managed cluster in Azure requires the following changes:

- The Azure CNI plugin version must be $\geq v1.0.27$. Older versions can have the wrong hardcoded Azure DNS servers.
- A recent-enough Kubernetes, with version $\geq 1.15.1$. This is required for Azure CNI.
- A recent-enough CMDaemon with version $\geq 8.2-138203$. This is required to handle Kubernetes $\geq 1.15.1$.
- Use of the `systemd-resolved` package instead of the `dnsmasq` package. This is because `azure-vnet` depends on `systemd-resolved`.
- A few custom iptables rules. These are defined in the kubelet service file.
- For Ubuntu:
  - A small change in the CoreDNS ConfigMap, to work around a known issue.
  - A patched `azure-vnet` binary, to support DNS servers not directly tied to specific network interfaces.

The steps to make Kubernetes for Azure work with Bright Cluster Manager according to these changes are described in sections 9.9.1–9.9.8.

Deployment of Kubernetes without those changes means that, while the nodes and pods would be up and running, inter-pod connectivity would fail.

The reason for failure is similar to the case of AWS as discussed in section 9.8. That is, IP addresses for the nodes hosting the Pods have the form 10.142.x.y, while a pod might be deployed with an IP address such as 172.29.0.2.

Source/destination checking cannot be disabled in Azure. Instead, to allow inter-pod connectivity in Bright Cluster Manager, the IP addresses are made to match.
A standard policy in Bright Cluster Manager is to not upgrade major software versions of the software packages used in a particular Bright Cluster Manager release. However, in order to accommodate Kubernetes on Azure in Bright Cluster Manager 8.2, a deviation from this policy occurs, and some extra packages have been introduced.

The steps to make Kubernetes for Azure work with Bright Cluster Manager are now described.

### 9.9.1 Updating CMDaemon

The CMDaemon package should be updated to the latest version. The minimal required version is: 8.2-138203. The requirement is for the regular nodes as well as for the head node, and should be fulfilled at least for all the nodes involved with the Kubernetes instance.

A failing `kubelet` service suggests that the CMDaemon is too old to support the newer Kubernetes version.

Typically, to update `cmdaemon` on the head node, the administrator runs, on the head node:

```bash
root@bright82 ~# yum update cmdaemon
```

Similarly, to update `cmdaemon` on the node images, the administrator runs:

```bash
root@bright82 ~]# chroot /cm/images/<node image>
root@bright82 [/]# yum update cmdaemon
...
root@bright82 [/]# exit
```

The updated image changes can be placed on the nodes themselves with the `imageupdate -w` command (section 5.6), or alternatively by simply rebooting the nodes.

### 9.9.2 Generating The Kubernetes Setup File `cm-kubernetes-setup.conf`

The Kubernetes setup run should be carried out (section 9.3.2) as usual, except that deployment step toward the end should not be carried out. Instead, the configuration should be saved and the wizard exited. The following properties can then be edited, as indicated by these sed one-liners, and assuming the configuration was saved in `cm-kubernetes-setup.conf`:

```bash
root@bright82 ~]# sed -i 's/cm-kubernetes-master$/cm-kubernetes-master15/g' cm-kubernetes-setup.conf
root@bright82 ~]# sed -i 's/cm-kubernetes-node$/cm-kubernetes-node15/g' cm-kubernetes-setup.conf
```

The one-liners change the packages selected, from the standard 8.2 versions, into enhanced versions that allow inter-pod connectivity. The lines that are altered are in the configuration overlay parts of the configuration file:

```bash
root@bright82 ~]# cat cm-kubernetes-setup.conf
...
    master:
        configuration_overlay:
            name: kube-default-master
            priority: 510
...
    packages:
    - cm-kubernetes-master15
    - cm-kubernetes-node15
...
```

With these changes in place, the new setup configuration can be deployed with:

```bash
root@bright82 ~]# cm-kubernetes-setup -c cm-kubernetes-setup.conf
```
### Secondary IP Address Configuration

A pool of IP addresses must be pre-allocated in the same virtual network. The pool is for the nodes that run pods. The IP addresses to be used are private IP addresses, not public IP addresses (figure 9.7).

![Figure 9.7: Assignment of secondary IP configurations to the NIC](image)

Allocation is possible via the Azure Portal web interface, but is not recommended, since it will be very slow. Instead, the `az` CLI utility, available at https://docs.microsoft.com/en-us/cli/azure/?view=azure-cli-latest, can be installed.

The most common use of the `az` utility is for commands such as:

**Example**

```bash
az login # login
az account list # read the hash for the Azure subscription ID you are interested in

export SUBSCRIPTION=<subscription ID>

az account set --subscription $SUBSCRIPTION
```

A session similar to the following can then be run, where some IP configurations are created, listed, and deleted:

**Example**

```bash
root@bright82 ~]# # also set the resource group of your cluster
root@bright82 ~]# export RESOURCE_GROUP=mycluster_cod_resource_group
root@bright82 ~]# # the network interface you wish to add IPs to
root@bright82 ~]# export NIC=cnode001-nic

root@bright82 ~]# # add IP configurations
```
9.9 Kubernetes On Azure

root@bright82 ~]# for i in {1..10}; do az network nic ip-config create -g $RESOURCE_GROUP --nic-name $NIC --name pod-$i; done

root@bright82 ~]# # list all IP configurations
root@bright82 ~]# az network nic ip-config list --resource-group $RESOURCE_GROUP --nic-name $NIC --output table

root@bright82 ~]# # delete
root@bright82 ~]# az network nic ip-config delete --resource-group $RESOURCE_GROUP --nic-name $NIC --name IPConfig-1

A handy script that can be used is a create.sh script, with an example usage of:
create.sh cnode001 15
which creates 15 IP addresses for cnode001.
The script itself is:

Example
#!/bin/bash
export RESOURCE_GROUP=mycluster_cod_resource_group
export NIC=${1:-cnode001}-nic
export NUM=${2:-10}
i=1
while [[ $i -le $NUM ]]; do
    az network nic ip-config create -g $RESOURCE_GROUP --nic-name $NIC --name pod-$i;
i=$((i + 1))
done

Similarly a delete.sh script can be used, with an example usage of:
delete.sh cnode001
which deletes the IP addresses allocated for cnode001:

Example
#!/bin/bash
export RESOURCE_GROUP=mycluster_cod_resource_group
export NIC=${1:-cnode001}-nic
while read config; do
    az network nic ip-config delete --resource-group $RESOURCE_GROUP --nic-name $NIC --name $config
done < <(az network nic ip-config list --resource-group $RESOURCE_GROUP --nic-name $NIC --output table|grep False | awk '{ print $1 }')

9.9.4 Installing Azure CNI
The CNI plugin can be installed by going into the software images used by the Kubernetes deployment using a chroot, and then running a session as follows:

root@bright82 ~]# wget https://raw.githubusercontent.com/Azure/azure-container-networking/master/scripts/install-cni-plugin.sh -O install-cni-plugin.sh
root@bright82 ~]# # replace the path in the script
root@bright82 ~]# sed -i 's/^CNI_BIN_DIR=.*$/CNI_BIN_DIR=/cm/local/apps/kubernetes/current/bin/cni/g'
root@bright82 ~]# bash install-cni-plugin.sh v1.0.27 v0.8.1

The preceding session places the installer script install-cni-plugin.sh in the image, and modifies the path to a specific one for the image.
A patched azure-vnet file can be picked up as follows:

© Bright Computing, Inc.
root@bright82 ~]# wget https://support2.brightcomputing.com/azure-k8s/azure-vnet -O
/cm/local/apps/kubernetes/current/bin/cni/azure-vnet

The upstream patch from Bright Computing can be seen at https://github.com/Azure/azure-container-networking/pull/419.

9.9.5 Installing Azure NPM
The azure-npm file should then be installed using this Yaml:

root@bright82 ~]# kubectl apply -f
https://raw.githubusercontent.com/Azure/azure-container-networking/master/npm/azure-npm.yaml

9.9.6 Ensuring Some Firewall Rules Are Applied Via Kubelet Service
Within the software images, the file /lib/systemd/system/kubelet.service should be edited to include the following two iptables lines. These are added before the ExecStart line:

ExecStartPre=/sbin/iptables -P FORWARD ACCEPT
ExecStartPre=/sbin/iptables -t nat -A POSTROUTING -m addrtype ! --dst-type local ! -d 10.142.0.0/16 -j MASQUERADE

In the preceding lines, the first `iptables` command enables connectivity from pods to the machine hosting the Pods. The second `iptables` command allows external connectivity from within the pods too.

The command `systemctl daemon-reload` should be run. In the chroot it prints an error, which should be ignored.

9.9.7 DNS Fix For Ubuntu Installations
For RHEL-based deployments this section can be skipped.

Some previous Ubuntu versions of Bright Cluster Manager 8.2 used dnsmasq instead of `system-resolved` for DNS services, as a workaround for upstream bugs in `system-resolved`. These were resolved in later releases.

Since Azure has a dependency on `system-resolved`, it means that the operating system must be moved over to use `systemd-resolved` if `dnsmasq` is being used.

A further step is to hardcode the DNS server. In theory this should not be needed, but there are some issues that appear in the Ubuntu distribution only at the time of writing (October 2019), due to the CoreDNS service not forwarding DNS server requests from the host to the CoreDNS pods.

root@bright82 ~]# cmsh
[bright82]% kubernetes
[bright82->kubernetes[default]]% appgroups
[bright82->kubernetes[default]->appgroups]% use system
[bright82->kubernetes[default]->appgroups[system]]% applications
[bright82->kubernetes[default]->appgroups[system]->applications]% use dns
[bright82->kubernetes[default]->appgroups[system]->applications[dns]]% set config

The ConfigMap for CoreDNS can then be changed:

... 
apiVersion: v1
kind: ConfigMap
metadata:
  name: coredns
  namespace: kube-system
data:
Corefile: |
.:53 {
  errors
  health
  kubernetes ${CMD_KUBE_DOMAIN} in-addr.arpa ip6.arpa {
    pods insecure
    upstream
    fallthrough in-addr.arpa ip6.arpa
  }
  prometheus :9153
  proxy . 10.142.255.254
  cache 30
  loop
  reload
  loadbalance
}

The only change is on the line proxy . 10.142.255.254. There, the entry /etc/resolv.conf has been replaced with the internal IP, 10.142.255.254, of the head node that is hosting the DNS server. After committing the change, CMDaemon should be allowed some time to update the changes. The command kubectl get cm -n kube-system coredns -o yaml lets the cluster administrator check for the change. When it is correct, the CoreDNS pods can be deleted with: kubectl delete pod -n kube-system -l k8s-app=kube-dns.

9.9.8 Restarting Services

Now all the compute nodes (worker nodes) that make up the Kubernetes instance should be rebooted, or an imageupdate -w should be run via cmsh for each of those nodes. A reboot is recommended because otherwise it may be extra work to ensure that all services are restarted, and that they are also started in the right order.

With both a reboot, or an imageupdate, the cluster administrator must delete the pods in order to allow the Ingress Controller to configure correctly when the pods are recreated automatically.

With a reboot, all pods automatically reinitialize with IP addresses from the 10.142.0.0/16 virtual network provided by the Azure CNI plugin. The pods can then be deleted to force recreation with the new IP addresses.

With an imageupdate the kubelet service must be restarted on all the nodes that are part of the Kubernetes instance. Then all the pods must be deleted, to force them to be recreated with new IP addresses.

Some examples:

Example

root@bright82 ~]$ kubectl delete pod -n kube-system --all
root@bright82 ~]$ kubectl delete pod -n ingress-nginx --all

The administrator should be aware of the possibility of “rogue pods”. These are pods that are not managed by a deployment or other higher level mechanism in Kubernetes which take care of immediately rescheduling deleted Pods.

In those cases a reboot is much safer. A gradual approach is also possible—it is not required to reboot all the nodes at once.
10

BeeGFS

10.1 BeeGFS Introduction

10.1.1 BeeGFS Concepts

BeeGFS is a high-performance parallel file system, developed by the Fraunhofer Competence Center for High Performance Computing, and optimized for intensive I/O. It uses a distributed metadata architecture designed for scalability and flexibility. More information about BeeGFS can be found at the official online documentation at https://www.beegfs.io/content/documentation/.

Bright Cluster Manager provides packages to allow BeeGFS to be deployed, managed, and monitored on a Bright cluster running RHEL or Centos. The deployment tool provided by Bright Cluster Manager is cm-beegfs-setup (section 10.2).

A BeeGFS cluster consists of a management server, one or more metadata servers and one or more storage servers. Clients should have client servers running on them.

If high-availability is required, then the administrator needs to arrange for it separately, for the BeeGFS management servers and Admon node.

Since BeeGFS is a parallel filesystem, it means that adding more nodes increases not only the storage capacity, but also system performance. BeeGFS can work with any local file system for data storage as long as it is POSIX compilant.

Versions of BeeGFS before 7.2 have an optional graphical user interface called Admon (section 10.3.4), which is integrated with Bright Cluster Manager. This has been dropped in version 7.2. The BeeGFS Mon monitoring service is on the roadmap for future Bright Cluster Manager integration.

10.1.2 BeeGFS Installation Notes And Options

By default all logging is done to the systemd journal. The installation logs can be viewed using:

    journalctl -u 'beegfs-*'.

If Admon is used, then it cannot use journalled logs by default. Logging into a file is needed instead. Log locations can be changed using cmsh or Bright View at any moment after installation, by editing BeeGFS roles for the corresponding configuration overlays (section 10.2.1). After the changes are made, the Admon service must be manually restarted on the Admon node (section 10.2.1):

Example

    systemctl restart beegfs-admon.service

Authentication is possible for the BeeGFS cluster via a shared secret file.

BeeGFS can be installed with these options:

- Admon (for versions of BeeGFS before 7.2)
- Shared secret file

© Bright Computing, Inc.
10.2 Deployment And Uninstallation Of BeeGFS With cm-beegfs-setup

Deployment and uninstallation for a Bright Cluster Manager can be carried out with the Ncurses-based cm-beegfs-setup utility. The utility is a part of a cluster-tools package that comes with Bright Cluster Manager. If the BeeGFS packages are not installed, then the utility installs them.

The cm-beegfs-setup utility can be run as root from the head node. The first Ncurses screen that shows up is the main menu for operations (figure 10.1)

![Figure 10.1: BeeGFS Script Main Menu](image)

In the main menu, the administrator can choose to:

- Deploy BeeGFS (section 10.2.1)
- Uninstall BeeGFS (section 10.2.2)
- Simply exit the Ncurses session

10.2.1 Deployment Of BeeGFS

If the Deploy option is chosen, then the deployment wizard starts.

**Configuration Overlays**

The wizard firsts asks for names for the configuration overlays (figure 10.2):

![Figure 10.2: Configuration Overlays Creation](image)

These overlay names are for:

- the BeeGFS metadata configuration overlay
- the BeeGFS storage configuration overlay
- the BeeGFS client configuration overlay

**Select Management Node**

The next screen (figure 10.3) then asks for a node to be selected to be the BeeGFS management server:
10.2 Deployment And Uninstallation Of BeeGFS With cm-beegfs-setup

Select Metadata Nodes

The next screen (figure 10.4) then asks for nodes to be selected to be the BeeGFS metadata servers:

Select Storage Nodes

The next screen (figure 10.5) asks for nodes to be selected to be the BeeGFS storage servers:

Select Client Nodes

The next screen (figure 10.6) asks for nodes to be selected to be the client nodes:
Please select client nodes for BeeGFS.

<table>
<thead>
<tr>
<th></th>
<th>t-wrk</th>
<th></th>
<th>(cores: 4, ram: 8 GB)</th>
</tr>
</thead>
<tbody>
<tr>
<td>[ ]</td>
<td>node001</td>
<td></td>
<td>category:default (cores: 2, ram: 4 GB)</td>
</tr>
<tr>
<td>[ ]</td>
<td>node002</td>
<td></td>
<td>category:default (cores: 2, ram: 4 GB)</td>
</tr>
<tr>
<td>[ ]</td>
<td>node003</td>
<td></td>
<td>category:default (cores: 2, ram: 4 GB)</td>
</tr>
<tr>
<td>[ ]</td>
<td>node004</td>
<td></td>
<td>category:default (cores: 2, ram: 4 GB)</td>
</tr>
<tr>
<td>[ ]</td>
<td>node005</td>
<td></td>
<td>category:default (cores: 2, ram: 4 GB)</td>
</tr>
</tbody>
</table>

Figure 10.6: Select Client Nodes

**Admon Deployment**
The next screen (figure 10.7) asks if the Admon GUI for BeeGFS is to be deployed:

Do you want to deploy Admon?

[ ] yes

[ ] no

Figure 10.7: Admon Deployment

**Select Admon Node**
If Admon installation is chosen, then the wizard asks (figure 10.8) for a node to be selected for Admon use:

Please select an admon node for BeeGFS.

<table>
<thead>
<tr>
<th></th>
<th>t-wrk</th>
<th></th>
<th>(cores: 4, ram: 8 GB)</th>
</tr>
</thead>
<tbody>
<tr>
<td>[ ]</td>
<td>node001</td>
<td></td>
<td>category:default (cores: 2, ram: 4 GB)</td>
</tr>
<tr>
<td>[ ]</td>
<td>node002</td>
<td></td>
<td>category:default (cores: 2, ram: 4 GB)</td>
</tr>
<tr>
<td>[ ]</td>
<td>node003</td>
<td></td>
<td>category:default (cores: 2, ram: 4 GB)</td>
</tr>
<tr>
<td>[ ]</td>
<td>node004</td>
<td></td>
<td>category:default (cores: 2, ram: 4 GB)</td>
</tr>
<tr>
<td>[ ]</td>
<td>node005</td>
<td></td>
<td>category:default (cores: 2, ram: 4 GB)</td>
</tr>
</tbody>
</table>

Figure 10.8: Select Admon Node

**Management Server Configuration**
The next screen (figure 10.9) asks for some configuration parameters for the BeeGFS management server to be set: 

© Bright Computing, Inc.
10.2 Deployment And Uninstallation Of BeeGFS With `cm-beegfs-setup`

The values required are for the:

- Path to a data directory
- Management server TCP port
- Management server UDP port

**Metadata Server Configuration**

The next screen (figure 10.10) asks for input on configuring the BeeGFS metadata servers:

The values required are for the:

- Path to a data directory
- Metadata server TCP port
- Metadata server UDP port

**Storage Server Configuration**

The next screen (figure 10.11) asks for input on setting some configuration parameters for the BeeGFS storage servers:

The items required are for the:

- Path to a data directory
- Storage server TCP port

© Bright Computing, Inc.
- Storage server UDP port

**Client Configuration**
The next screen (figure 10.12) asks for input on configuring the BeeGFS clients:

![Figure 10.12: Client Configuration](image)

The inputs required are for the:
- Client UDP port
- Helper TCP port

**Admon Configuration**
If Admon has been chosen for installation, then the wizard asks (figure 10.13) for some configuration settings for Admon:

![Figure 10.13: Admon Configuration](image)

The input values required are:
- Path to the database file
- Admon HTTP port
- Admon UDP port

After installation is complete, the Admon jar file can be downloaded (section 10.3.4).

**Authentication Configuration**
In the next screen, (figure 10.14) a path must be specified for the shared secret file:

![Figure 10.14: Authentication Configuration](image)

This is a file with arbitrary content. If it is set, then it must exist on all BeeGFS nodes.

© Bright Computing, Inc.
10.3 Managing The Deployed BeeGFS Instance

Deployment Summary

Finally, the wizard displays a deployment summary (figure 10.15):

![Deployment Summary](image)

Figure 10.15: Deployment Summary

From this screen the deployment process can be started, the deployment configuration can be viewed and saved, or the wizard can simply be exited.

Command Line Installation

If a configuration file is saved from a `cm-beegfs-setup` run, then it can be used to install a BeeGFS cluster automatically:

**Example**

```bash
cm-beegfs-setup -c cm-beegfs-setup.conf
```

10.2.2 Uninstalling BeeGFS

If the Uninstall option is chosen from the main menu (figure 10.1), then the uninstallation process starts.

An extra *Are you sure?* prompt is thrown in the way first, to help avoid a mistaken removal. After confirmation, a BeeGFS instance must then be selected for the removal, by selecting its management node (figure 10.16):

**Uninstall Management Node Selection**

![Uninstall Management Node Selection](image)

Figure 10.16: Uninstall Management Node Selection

The uninstallation process then starts. It can take several minutes in order to ensure that services are stopped properly, so that errors and junk files are avoided.

10.3 Managing The Deployed BeeGFS Instance

A BeeGFS cluster consists of:

- One Management node
- Some Metadata nodes
- Some Storage nodes

© Bright Computing, Inc.
Information about the right number of nodes to use can be found at https://www.beegfs.io/docs/whitewpapers/Picking_the_right_Number_of_Targets_per_Server_for_BeeGFS_by_ThinkParQ.pdf

BeeGFS clients should have client servers running on them.

### 10.3.1 Setup
Associated with BeeGFS are the following BeeGFS-related packages, most of which provide services:

1. beegfs-mgmtd
2. beegfs-meta
3. beegfs-storage
4. beegfs-client
5. beegfs-helperd
6. beegfs-utils
7. beegfs-admon

The repository from which these packages are picked up is added to the repositories list when BeeGFS is installed by Bright Cluster Manager (section 10.2), and the packages are then installed on all of the cluster nodes. The services are assignable by the cluster administrator to cluster devices as roles. In cmsh these BeeGFS roles are treated as objects, and the configuration of these roles is determined by role parameters (object attributes). The roles are described in detail section 10.3.2.

### 10.3.2 BeeGFS Objects

#### BeeGFSManagement Role
The BeeGFSManagement role is used to configure BeeGFS management services. The role parameters are described in table 10.1:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to beegfs-mgmtd</th>
</tr>
</thead>
<tbody>
<tr>
<td>Data directory</td>
<td>Path to data directory</td>
<td>storeMgmtdDirectory</td>
</tr>
<tr>
<td>Allow new servers</td>
<td>Allow new servers registration</td>
<td>sysAllowNewServers</td>
</tr>
<tr>
<td>Allow new storage targets</td>
<td>Allow new storage targets registration</td>
<td>sysAllowNewTargets</td>
</tr>
<tr>
<td>Authentification file</td>
<td>Path to shared secret authentication file</td>
<td>connAuthFile</td>
</tr>
<tr>
<td>Backlog TCP</td>
<td>TCP listen backlog size</td>
<td>connBacklogTCP</td>
</tr>
</tbody>
</table>

...continues
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to beegfs-mgmd</th>
</tr>
</thead>
<tbody>
<tr>
<td>Interfaces file</td>
<td>Path to file with interfaces list</td>
<td>connInterfacesFile</td>
</tr>
<tr>
<td>Management port TCP</td>
<td>TCP port for management service</td>
<td>connMgmtdPortTCP</td>
</tr>
<tr>
<td>Management port UDP</td>
<td>UDP port for management service</td>
<td>connMgmtdPortUDP</td>
</tr>
<tr>
<td>Net filter file</td>
<td>Path to file with list of allowed IP subnets</td>
<td>connNetFilterFile</td>
</tr>
<tr>
<td>Log level</td>
<td>Log level</td>
<td>logLevel</td>
</tr>
<tr>
<td>No log date</td>
<td>Do not show date along with time in log</td>
<td>logNoDate</td>
</tr>
<tr>
<td>Number of log lines</td>
<td>Number of lines in log file, after which it will be rotated</td>
<td>logNumLines</td>
</tr>
<tr>
<td>Number of log rotated files</td>
<td>Number of old log files to keep</td>
<td>logNumRotatedFiles</td>
</tr>
<tr>
<td>Log file</td>
<td>Path to log file, empty means logs go to the journal</td>
<td>logStdFile</td>
</tr>
<tr>
<td>Enable Quota</td>
<td>Enable quota</td>
<td>quotaEnableEnforcement</td>
</tr>
<tr>
<td>Quota query GID file</td>
<td>Path to file with GIDs to be checked by quota</td>
<td>quotaQueryGIDFile</td>
</tr>
<tr>
<td>Quota query GID range</td>
<td>GID range to be checked by quota</td>
<td>quotaQueryGIDRange</td>
</tr>
<tr>
<td>Quota query UID file</td>
<td>Path to file with UIDs to be checked by quota</td>
<td>quotaQueryUIDFile</td>
</tr>
<tr>
<td>Quota query UID range</td>
<td>UID range to be checked by quota</td>
<td>quotaQueryUIDRange</td>
</tr>
<tr>
<td>Quota query type</td>
<td>Query type for quota</td>
<td>quotaQueryType</td>
</tr>
</tbody>
</table>

...continues
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to beegfs-mgtd</th>
</tr>
</thead>
<tbody>
<tr>
<td>Quota query with system users/groups</td>
<td>Allow also system users/groups to be checked by quota</td>
<td>quotaQueryWithSystemUsersGroups</td>
</tr>
<tr>
<td>Quota update interval</td>
<td>Quota update interval</td>
<td>quotaUpdateIntervalMin</td>
</tr>
<tr>
<td>Target offline timeout</td>
<td>Timeout for targets on storage server to be considered offline when no target status received</td>
<td>sysTargetOfflineTimeoutSecs</td>
</tr>
<tr>
<td>Client auto remove time</td>
<td>Time after which an unreachable node is considered dead</td>
<td>tuneClientAutoRemoveMins</td>
</tr>
<tr>
<td>Number of workers</td>
<td>Number of worker threads</td>
<td>tuneNumWorkers</td>
</tr>
<tr>
<td>Meta dynamic pools</td>
<td>Raise lower limits if difference in capacity becomes too large between targets</td>
<td>tuneMetaDynamicPools</td>
</tr>
<tr>
<td>Meta inodes low limit</td>
<td>Metadata inode free space pool threshold</td>
<td>tuneMetaInodesLowLimit</td>
</tr>
<tr>
<td>Meta inodes emergency limit</td>
<td>Metadata inode free space pool threshold</td>
<td>tuneMetaInodesEmergencyLimit</td>
</tr>
<tr>
<td>Meta space low limit</td>
<td>Metadata space free space pool threshold</td>
<td>tuneMetaSpaceLowLimit</td>
</tr>
<tr>
<td>Meta space emergency limit</td>
<td>Metadata space free space pool threshold</td>
<td>tuneMetaSpaceEmergencyLimit</td>
</tr>
<tr>
<td>Storage dynamic pools</td>
<td>Raise lower limits if difference in capacity becomes too large between targets</td>
<td>tuneStorageDynamicPools</td>
</tr>
<tr>
<td>Storage inodes low limit</td>
<td>Storage inode free space pool threshold</td>
<td>tuneStorageInodesLowLimit</td>
</tr>
<tr>
<td>Storage inodes emergency limit</td>
<td>Storage inode free space pool threshold</td>
<td>tuneStorageInodesEmergencyLimit</td>
</tr>
</tbody>
</table>

...continues
### 10.3 Managing The Deployed BeeGFS Instance

...continued

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to beegfs-mgmtd</th>
</tr>
</thead>
<tbody>
<tr>
<td>Storage space low limit</td>
<td>Storage space free space pool threshold</td>
<td>tuneStorageSpaceLowLimit</td>
</tr>
<tr>
<td>Storage space emergency limit</td>
<td>Storage space free space pool threshold</td>
<td>tuneStorageSpaceEmergencyLimit</td>
</tr>
</tbody>
</table>

| Table 10.1: BeeGFSManagement role parameters and beegfs-mgmtd options |

The beegfs-mgmtd settings are updated by Bright Cluster Manager in `/etc/beegfs/beegfs-mgmtd.conf`. The settings can be managed in cmsh via the `beegfs::management` role. Roles are, as usual, assigned via device, category or configurationoverlay mode. The parameters of the role that correspond to the parameters in the preceding table can then be set.

**Example**

```
[bright82 ~]# cmsh
[bright82%] device roles node001
[bright82->device[node001]->roles]% assign bee<TAB><TAB><TAB>
beegfs::admon beegfs::client beegfs::management beegfs::metadata beegfs::storage
[bright82->device[node001]->roles]% assign beegfs::management
[bright82->device[node001]->roles[BeeGFS::Management]]% show
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add services</td>
<td>yes</td>
</tr>
<tr>
<td>Name</td>
<td>BeeGFS::Management</td>
</tr>
<tr>
<td>Provisioning associations</td>
<td>&lt;0 internally used&gt;</td>
</tr>
<tr>
<td>Revision</td>
<td>BeeGFSManagementRole</td>
</tr>
<tr>
<td>Allow new servers</td>
<td>yes</td>
</tr>
<tr>
<td>Data directory</td>
<td><code>/var/lib/beegfs/management</code></td>
</tr>
<tr>
<td>Allow new storage targets</td>
<td>yes</td>
</tr>
<tr>
<td>Authentication file</td>
<td></td>
</tr>
<tr>
<td>Backlog TCP</td>
<td>128</td>
</tr>
<tr>
<td>Interfaces file</td>
<td></td>
</tr>
<tr>
<td>Management port TCP</td>
<td>8008</td>
</tr>
<tr>
<td>Management port UDP</td>
<td>8008</td>
</tr>
<tr>
<td>Net filter file</td>
<td></td>
</tr>
<tr>
<td>Log file</td>
<td></td>
</tr>
<tr>
<td>Log level</td>
<td>2</td>
</tr>
<tr>
<td>No log date</td>
<td>no</td>
</tr>
<tr>
<td>Number of log lines</td>
<td>50000</td>
</tr>
<tr>
<td>Number of log rotated files</td>
<td>5</td>
</tr>
<tr>
<td>Enable Quota</td>
<td>no</td>
</tr>
<tr>
<td>Quota query GID file</td>
<td></td>
</tr>
<tr>
<td>Quota query GID range</td>
<td></td>
</tr>
<tr>
<td>Quota query UID file</td>
<td></td>
</tr>
<tr>
<td>Quota query UID range</td>
<td></td>
</tr>
<tr>
<td>Quota query type</td>
<td>system</td>
</tr>
<tr>
<td>Quota query with system users groups</td>
<td>no</td>
</tr>
<tr>
<td>Quota update interval</td>
<td>10m</td>
</tr>
</tbody>
</table>

© Bright Computing, Inc.
Target offline timeout 3m
Client auto remove time 30m
Number of workers 4
Meta dynamic pools yes
Meta inodes emergency limit 1M
Meta inodes low limit 10M
Meta space emergency limit 3G
Meta space low limit 10G
Storage dynamic pools yes
Storage inodes emergency limit 1M
Storage inodes low limit 10M
Storage space emergency limit 20G
Storage space low limit 1T

BeeGFSMetadata Role
The BeeGFSMetadata role is used for configuring BeeGFS metadata services. The role parameters are described in Table 10.2:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to beegfs-meta</th>
</tr>
</thead>
<tbody>
<tr>
<td>Data directory</td>
<td>Path to the data directory</td>
<td>storeMetaDirectory</td>
</tr>
<tr>
<td>Management node</td>
<td>Node with a management role</td>
<td>sysMgmtdHost</td>
</tr>
<tr>
<td>Authentication file</td>
<td>Path to the shared secret</td>
<td>connAuthFile</td>
</tr>
<tr>
<td>Backlog TCP</td>
<td>TCP listen backlog</td>
<td>connBacklogTCP</td>
</tr>
<tr>
<td>Fallback expiration</td>
<td>Time after which a connection to a fallback interface expires</td>
<td>connFallbackExpirationSecs</td>
</tr>
<tr>
<td>Interfaces file</td>
<td>Path to the file with a list of interfaces</td>
<td>connInterfacesFile</td>
</tr>
<tr>
<td>Max internode number</td>
<td>Max number of simultaneous connections to the same node</td>
<td>connMaxInternodeNum</td>
</tr>
<tr>
<td>Meta port TCP</td>
<td>TCP port for metadata service</td>
<td>connMetaPortTCP</td>
</tr>
<tr>
<td>Meta port UDP</td>
<td>UDP port for metadata service</td>
<td>connMetaPortUDP</td>
</tr>
</tbody>
</table>

...continues
### 10.3 Managing The Deployed BeeGFS Instance

---continued---

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to \texttt{beegfs-meta}</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Net filter file</strong></td>
<td>Path to a file with a list of allowed IP subnets</td>
<td>\texttt{connNetFilterFile}</td>
</tr>
<tr>
<td><strong>Use RDMA</strong></td>
<td>Use RDMA</td>
<td>\texttt{connUseRDMA}</td>
</tr>
<tr>
<td><strong>RDMA type of service</strong></td>
<td>RDMA type of service</td>
<td>\texttt{connRDMATypeOfService}</td>
</tr>
<tr>
<td><strong>Tcp only filter file</strong></td>
<td>Path to a file with a list of no DRMA IP ranges</td>
<td>\texttt{connTcpOnlyFilterFile}</td>
</tr>
<tr>
<td><strong>Log level</strong></td>
<td>Log level</td>
<td>\texttt{logLevel}</td>
</tr>
<tr>
<td><strong>No log date</strong></td>
<td>Do not show date along with time in log</td>
<td>\texttt{logNoDate}</td>
</tr>
<tr>
<td><strong>Number of log lines</strong></td>
<td>Number of lines in log file, after which it will be rotated</td>
<td>\texttt{logNumLines}</td>
</tr>
<tr>
<td><strong>Number of log rotated files</strong></td>
<td>Number of old log files to keep</td>
<td>\texttt{logNumRotatedFiles}</td>
</tr>
<tr>
<td><strong>Log file</strong></td>
<td>Path to the log file, empty means logs go to the journal</td>
<td>\texttt{logStdFile}</td>
</tr>
<tr>
<td><strong>Client xattrs</strong></td>
<td>Enable client-side extended attributes</td>
<td>\texttt{storeClientXAttrs}</td>
</tr>
<tr>
<td><strong>Client ACLs</strong></td>
<td>Enable handling and storage of client-side ACLs</td>
<td>\texttt{storeClientACLs}</td>
</tr>
<tr>
<td><strong>Use extended attributes</strong></td>
<td>Store metadata as extended attributes or not</td>
<td>\texttt{storeUseExtendedAttributes}</td>
</tr>
<tr>
<td><strong>Target attachment file</strong></td>
<td>File with a list of targets to be grouped within the same domain for randominternode</td>
<td>\texttt{sysTargetAttachmentFile}</td>
</tr>
<tr>
<td><strong>Target offline timeout</strong></td>
<td>Timeout for targets on a storage server to be considered offline when no target status is received</td>
<td>\texttt{sysTargetOfflineTimeoutSecs}</td>
</tr>
</tbody>
</table>

---continues---

© Bright Computing, Inc.
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to beegfs-meta</th>
</tr>
</thead>
<tbody>
<tr>
<td>Allow user set pattern</td>
<td>Allow non-privileged users to modify stripe pattern settings for directories they own</td>
<td>sysAllowUserSetPattern</td>
</tr>
<tr>
<td>Bind to numa zone</td>
<td>Zero-based NUMA zone number to which all threads of metadata process should be bound</td>
<td>tuneBindToNumaZone</td>
</tr>
<tr>
<td>Number of stream listeners</td>
<td>The number of threads waiting for incoming data events</td>
<td>tuneNumStreamListeners</td>
</tr>
<tr>
<td>Number of workers</td>
<td>Number of worker threads</td>
<td>tuneNumWorkers</td>
</tr>
<tr>
<td>Target chooser</td>
<td>The algorithm to choose storage targets for file creation</td>
<td>tuneTargetChooser</td>
</tr>
<tr>
<td>Use aggressive stream poll</td>
<td>Actively poll for events instead of sleeping until an event occurs</td>
<td>tuneUseAggressiveStreamPoll</td>
</tr>
<tr>
<td>Use per user msg queues</td>
<td>Use per-user queues for pending requests</td>
<td>tuneUsePerUserMsgQueues</td>
</tr>
</tbody>
</table>

Table 10.2: BeeGFSMetadata role parameters and beegfs-meta options

The beegfs-meta settings are updated by Bright Cluster Manager in `/etc/beegfs/beegfs-meta.conf`. The settings can be managed in cmsh via the beegfs::metadata role. Roles are, as usual, assigned via device, category or configurationoverlay mode. The parameters of the role that correspond to the parameters in the preceding table can then be set.

**Example**

```
[bright82 ~]# cmsh
[bright82] category roles default
[bright82->category[default]->roles]% assign bee<TAB><TAB><TAB>
beegfs::admon  beegfs::client  beegfs::management  beegfs::metadata  beegfs::storage
[bright82->category[default]->roles]% assign beegfs::metadata
[bright82->category[default]->roles][BeeGFS::Metadata%] show
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add services</td>
<td>yes</td>
</tr>
<tr>
<td>Name</td>
<td>BeeGFS::Metadata</td>
</tr>
<tr>
<td>Provisioning assoc.</td>
<td>&lt;0 internally used&gt;</td>
</tr>
<tr>
<td>Revision</td>
<td>BeeGFSMetadataRole</td>
</tr>
<tr>
<td>Data directory</td>
<td>/var/lib/beegfs/metadata</td>
</tr>
<tr>
<td>Management node</td>
<td></td>
</tr>
</tbody>
</table>

© Bright Computing, Inc.
### BeeGFSStorage Role

The BeeGFSStorage role is used for configuring BeeGFS storage services. The role parameters are described in Table 10.3:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to beegfs-storage</th>
</tr>
</thead>
<tbody>
<tr>
<td>Data directories</td>
<td>Path to the data directories</td>
<td>storeStorageDirectory</td>
</tr>
<tr>
<td>Management node</td>
<td>Node with a management role</td>
<td>sysMgmtdHost</td>
</tr>
<tr>
<td>Authentification file</td>
<td>Path to the shared secret file</td>
<td>connAuthFile</td>
</tr>
<tr>
<td>Backlog TCP</td>
<td>TCP listen backlog</td>
<td>connBacklogTCP</td>
</tr>
<tr>
<td>Interfaces file</td>
<td>Path to the file with a list of interfaces for communication</td>
<td>connInterfacesFile</td>
</tr>
</tbody>
</table>

...continues
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to beegfs-storage</th>
</tr>
</thead>
<tbody>
<tr>
<td>Max internode number</td>
<td>Max number of simultaneous connections to the same node</td>
<td>connMaxInternodeNum</td>
</tr>
<tr>
<td>Storage port TCP</td>
<td>TCP port for storage service</td>
<td>connStoragePortTCP</td>
</tr>
<tr>
<td>Storage port UDP</td>
<td>UDP port for storage service</td>
<td>connStoragePortUDP</td>
</tr>
<tr>
<td>Net filter file</td>
<td>Path to a file with a list of allowed IP subnets</td>
<td>connNetFilterFile</td>
</tr>
<tr>
<td>Use RDMA</td>
<td>Use RDMA</td>
<td>connUseRDMA</td>
</tr>
<tr>
<td>RDMA type of service</td>
<td>RDMA type of service</td>
<td>connRDMATypeOfService</td>
</tr>
<tr>
<td>Tcp only filter file</td>
<td>Path to a file with a list of subnets with no RDMA</td>
<td>connTcpOnlyFilterFile</td>
</tr>
<tr>
<td>Log level</td>
<td>Log level</td>
<td>logLevel</td>
</tr>
<tr>
<td>No log date</td>
<td>Do not show date along with time in log</td>
<td>logNoDate</td>
</tr>
<tr>
<td>Number of log lines</td>
<td>Number of lines in log file, after which it will be rotated</td>
<td>logNumLines</td>
</tr>
<tr>
<td>Number of log rotated files</td>
<td>Number of old log files to keep</td>
<td>logNumRotatedFiles</td>
</tr>
<tr>
<td>Log file</td>
<td>Path to the log file, empty means logs go to the journal</td>
<td>logStdFile</td>
</tr>
<tr>
<td>Resync safety threshold</td>
<td>Add an extra amount of time to the last successful communication timestamp in case of a potential cache loss</td>
<td>sysResyncSafetyThresholdMins</td>
</tr>
<tr>
<td>Target offline timeout</td>
<td>Timeout until targets on a storage server are considered offline when no target status is received</td>
<td>sysTargetOfflineTimeoutSecs</td>
</tr>
</tbody>
</table>
### Parameter Description Option to beegfs-storage

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to beegfs-storage</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bind to numa zone</td>
<td>Zero-based NUMA zone number to which all threads of metadata process should be bound</td>
<td>tuneBindToNumaZone</td>
</tr>
<tr>
<td>File read ahead size</td>
<td>Byte range submitted to the kernel for read-ahead after number of bytes was already read from target</td>
<td>tuneFileReadAheadSize</td>
</tr>
<tr>
<td>File read ahead trigger size</td>
<td>Number of bytes read after which the read-ahead is triggered</td>
<td>tuneFileReadAheadTriggerSize</td>
</tr>
<tr>
<td>File read size</td>
<td>Maximum amount of data server should read in a single operation</td>
<td>tuneFileReadSize</td>
</tr>
<tr>
<td>File write size</td>
<td>Maximum amount of data server should write in a single operation</td>
<td>tuneFileWriteSize</td>
</tr>
<tr>
<td>File write sync size</td>
<td>Number of bytes after which kernel advised to commit data</td>
<td>tuneFileWriteSyncSize</td>
</tr>
<tr>
<td>Number of resync gather slaves</td>
<td>Number of threads to gather filesystem information for a buddy mirror resync</td>
<td>tuneNumResyncGatherSlaves</td>
</tr>
<tr>
<td>Number of resync slaves</td>
<td>Number of threads to sync filesystem information for a buddy mirror resync</td>
<td>tuneNumResyncSlaves</td>
</tr>
<tr>
<td>Number of stream listeners</td>
<td>Number of threads waiting for incoming data events</td>
<td>tuneNumStreamListeners</td>
</tr>
<tr>
<td>Number of workers</td>
<td>Number of worker threads</td>
<td>tuneNumWorkers</td>
</tr>
<tr>
<td>Use aggressive stream poll</td>
<td>Actively poll for events instead of sleeping until an event occur</td>
<td>tuneUseAggressiveStreamPoll</td>
</tr>
<tr>
<td>Use per target workers</td>
<td>Create a separate set of workers and attach it for each storage target</td>
<td>tuneUsePerTargetWorkers</td>
</tr>
<tr>
<td>Use per user msg queues</td>
<td>Use per-user queues for pending requests</td>
<td>tuneUsePerUserMsgQueues</td>
</tr>
</tbody>
</table>

...continues
Worker buffer size | Size of network and io buffers, allocated for each worker | `tuneWorkerBufSize`

**Table 10.3: BeeGFSStorage role parameters and beegfs-storage options**

The beegfs-storage settings are updated by Bright Cluster Manager in `/etc/beegfs/beegfs-storage.conf`.

The settings can be managed in cmsh via the `beegfs::storage` role. Roles are, as usual, assigned via device, category or configurationoverlay mode. The parameters of the role that correspond to the parameters in the preceding table can then be set.

**Example**

```
[bright82 ~]$ cmsh
[bright82]% category roles default
[bright82->category[default]->roles]% assign bee<TAB><TAB><TAB>
beegfs::admon  beegfs::client  beegfs::management  beegfs::metadata  beegfs::storage
[bright82->category[default]->roles]% assign beegfs::storage
[bright82->category[default]->roles[BeegFS::Storage]]% show
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add services</td>
<td>yes</td>
</tr>
<tr>
<td>Name</td>
<td>BeeGFS::Storage</td>
</tr>
<tr>
<td>Provisioning associations</td>
<td>&lt;0 internally used&gt;</td>
</tr>
<tr>
<td>Revision</td>
<td></td>
</tr>
<tr>
<td>Type</td>
<td>BeeGFSStorageRole</td>
</tr>
<tr>
<td>Data directories</td>
<td>/var/lib/beegfs/storage</td>
</tr>
<tr>
<td>Management node</td>
<td></td>
</tr>
<tr>
<td>Authentication file</td>
<td></td>
</tr>
<tr>
<td>Backlog TCP</td>
<td>128</td>
</tr>
<tr>
<td>Interfaces file</td>
<td></td>
</tr>
<tr>
<td>Max internode number</td>
<td>12</td>
</tr>
<tr>
<td>Net filter file</td>
<td></td>
</tr>
<tr>
<td>RDMA type of service</td>
<td>0</td>
</tr>
<tr>
<td>Storage port TCP</td>
<td>8003</td>
</tr>
<tr>
<td>Storage port UDP</td>
<td>8003</td>
</tr>
<tr>
<td>Tcp only filter file</td>
<td></td>
</tr>
<tr>
<td>Use RDMA</td>
<td>yes</td>
</tr>
<tr>
<td>Log file</td>
<td></td>
</tr>
<tr>
<td>Log level</td>
<td>3</td>
</tr>
<tr>
<td>No log date</td>
<td>no</td>
</tr>
<tr>
<td>Number of log lines</td>
<td>50000</td>
</tr>
<tr>
<td>Number of log rotated files</td>
<td>5</td>
</tr>
<tr>
<td>Resync safety threshold</td>
<td>10m</td>
</tr>
<tr>
<td>Target offline timeout</td>
<td>3m</td>
</tr>
<tr>
<td>Bind to numa zone</td>
<td></td>
</tr>
<tr>
<td>File read ahead size</td>
<td>0m</td>
</tr>
<tr>
<td>File read ahead trigger size</td>
<td>4m</td>
</tr>
<tr>
<td>File read size</td>
<td>128k</td>
</tr>
<tr>
<td>File write size</td>
<td>128k</td>
</tr>
<tr>
<td>File write sync size</td>
<td>0m</td>
</tr>
<tr>
<td>Number of resync gather slaves</td>
<td>6</td>
</tr>
<tr>
<td>Number of resync slaves</td>
<td>12</td>
</tr>
</tbody>
</table>
10.3 Managing The Deployed BeeGFS Instance

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to beegfs-admon</th>
</tr>
</thead>
<tbody>
<tr>
<td>Database file</td>
<td>Path to the database</td>
<td>databaseFile</td>
</tr>
<tr>
<td>Management node</td>
<td>Node with a management role</td>
<td>sysMgmtdHost</td>
</tr>
<tr>
<td>Clear database</td>
<td>Delete database on startup</td>
<td>clearDatabase</td>
</tr>
<tr>
<td>Query interval</td>
<td>Interval for querying servers for new statistics data</td>
<td>queryInterval</td>
</tr>
<tr>
<td>Authentication file</td>
<td>Path to the shared secret authentification file</td>
<td>connAuthFile</td>
</tr>
<tr>
<td>Fallback expiration</td>
<td>Time after which a connection to a fallback interface expires</td>
<td>connFallbackExpirationSecs</td>
</tr>
<tr>
<td>Interfaces file</td>
<td>Path to the file with a list of interfaces for communication</td>
<td>connInterfacesFile</td>
</tr>
<tr>
<td>Max internode number</td>
<td>Max number of simultaneous connections to the same node</td>
<td>connMaxInternodeNum</td>
</tr>
<tr>
<td>HTTP port</td>
<td>TCP port for built-in web server</td>
<td>httpPort</td>
</tr>
<tr>
<td>Admon port UDP</td>
<td>UDP port for admon service</td>
<td>connAdmonPortUDP</td>
</tr>
<tr>
<td>Net filter file</td>
<td>Path to a file with a list of allowed IP subnets</td>
<td>connNetFilterFile</td>
</tr>
</tbody>
</table>

...continues

BeeGFSAdmon Role
The BeeGFSAdmon role is used for configuring BeeGFS admon services. The role parameters are described in Table 10.4:
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to beegfs-admon</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tcp only filter file</td>
<td>Path to a file with a list of no RDMA IP ranges</td>
<td>connTcpOnlyFilterFile</td>
</tr>
<tr>
<td>Log level</td>
<td>Log level</td>
<td>logLevel</td>
</tr>
<tr>
<td>No log date</td>
<td>Do not show date along with time in log</td>
<td>logNoDate</td>
</tr>
<tr>
<td>Number of log lines</td>
<td>Number of lines in log file, after which it will be rotated</td>
<td>logNumLines</td>
</tr>
<tr>
<td>Number of log rotated files</td>
<td>Number of old log files to keep</td>
<td>logNumRotatedFiles</td>
</tr>
<tr>
<td>Log file</td>
<td>Path to the log file, empty means logs go to the journal</td>
<td>logStdFile</td>
</tr>
<tr>
<td>Enable mail</td>
<td>Enable email notification</td>
<td>mailEnabled</td>
</tr>
<tr>
<td>Smtp send type</td>
<td>How to send the email to an SMTP server</td>
<td>mailSmtpSendType</td>
</tr>
<tr>
<td>Sendmail path</td>
<td>Path to the sendmail binary</td>
<td>mailSendmailPath</td>
</tr>
<tr>
<td>Check interval time</td>
<td>Interval for checking for new events that must be mailed</td>
<td>mailCheckIntervalTimeSec</td>
</tr>
<tr>
<td>Minimal down time</td>
<td>Interval of node downtime after which it should be reported</td>
<td>mailMinDownTimeSec</td>
</tr>
<tr>
<td>Mail recipients</td>
<td>Email addresses to send notifications to</td>
<td>mailRecipient</td>
</tr>
<tr>
<td>Resend mail time</td>
<td>Time after which to send email again as a reminder</td>
<td>mailResendMailTimeMin</td>
</tr>
<tr>
<td>Mail sender</td>
<td>Email notification from address</td>
<td>mailSender</td>
</tr>
<tr>
<td>SMTP server</td>
<td>SMTP server to use for sending emails</td>
<td>mailSmtpServer</td>
</tr>
</tbody>
</table>

...continues
10.3 Managing The Deployed BeeGFS Instance

...continued

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to beegfs-admon</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of workers</td>
<td>Number of worker threads</td>
<td>tuneNumWorkers</td>
</tr>
</tbody>
</table>

Table 10.4: BeeGFSAdmon role parameters and beegfs-admon options

The beegfs-admon settings are updated by Bright Cluster Manager in `/etc/beegfs/beegfs-admon.conf`. The settings can be managed in cmsh via the beegfs::admon role:

Example

```
[bright82 ~]# cmsh
[bright82]% category roles default
[bright82->category[default]->roles]% assign bee<TAB><TAB><TAB>
beegfs::admon beegfs::client beegfs::management beegfs::metadata beegfs::storage
[bright82->category[default]->roles]% assign beegfs::admon
[bright82->category[default]->roles][BeeGFS::Admon]% show
```

Parameter          | Value                                                                 |
---------------------------------------------|-----------------------------------------------------------------------|
Add services       | yes                                                                   |
Name               | BeeGFS::Admon                                                         |
Provisioning associations | <0 internally used>                                                  |
Revision           | BeeGFSAdmonRole                                                      |
Database file      | /var/lib/beegfs/beegfs-admon.db                                      |
Management node    |                                                                       |
Clear database     | no                                                                    |
Query interval     | 5s                                                                    |
Admon port UDP     | 8007                                                                 |
Authentication file|                                                                       |
Fallback expiration| 15m                                                                   |
HTTP port          | 8000                                                                 |
Interfaces file    |                                                                       |
Max internode number | 3                                                                    |
Net filter file    |                                                                       |
Tcp only filter file|                                                                       |
Log file           |                                                                       |
Log level          | 3                                                                    |
No log date        | no                                                                    |
Number of log lines| 50000                                                                |
Number of log rotated files | 5                                                                 |
Check interval time| 30s                                                                   |
Enable mail        | no                                                                    |
Mail recipients    |                                                                       |
Mail sender        |                                                                       |
Minimal down time  | 10s                                                                   |
Resend mail time   | 1h                                                                    |
SMTP server        |                                                                       |
Sendmail path      | sendmail                                                              |
Smtp send type     | socket                                                                |
Number of workers  | 4                                                                    |

BeeGFSClient Role
The BeeGFSClient role is used for configuring BeeGFS client and helperd services.

© Bright Computing, Inc.
The role parameters are described in table 10.5 and in table 10.6:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to beegfs-client</th>
</tr>
</thead>
<tbody>
<tr>
<td>Management node</td>
<td>Node with a management role</td>
<td>sysMgmtdHost</td>
</tr>
<tr>
<td>Authentification file</td>
<td>Path to the shared secret authentification file</td>
<td>connAuthFile</td>
</tr>
<tr>
<td>Client port UDP</td>
<td>UDP port for client service</td>
<td>connClientPortUDP</td>
</tr>
<tr>
<td>Helper port TCP</td>
<td>TCP port for helper service</td>
<td>connHelperdPortTCP</td>
</tr>
<tr>
<td>Communication retry time</td>
<td>Time for retries in case of a network failure</td>
<td>connCommRetrySecs</td>
</tr>
<tr>
<td>Fallback expiration time</td>
<td>Time after which a connection to a fallback interface expires</td>
<td>connFallbackExpirationSecs</td>
</tr>
<tr>
<td>Interfaces file</td>
<td>Path to file with list of communication interfaces</td>
<td>connInterfacesFile</td>
</tr>
<tr>
<td>Max internode number</td>
<td>Maximum simultaneous connections to the same node</td>
<td>connMaxInternodeNum</td>
</tr>
<tr>
<td>Net filter file</td>
<td>Path to a file with a list of allowed IP subnets</td>
<td>connNetFilterFile</td>
</tr>
<tr>
<td>Use RDMA</td>
<td>Use RDMA</td>
<td>connUseRDMA</td>
</tr>
<tr>
<td>RDMA buffers number</td>
<td>Number of RDMA buffers</td>
<td>connRDMABufNum</td>
</tr>
<tr>
<td>RDMA buffer size</td>
<td>Maximum size of a buffer that will be sent over the network</td>
<td>connRDMABufSize</td>
</tr>
<tr>
<td>RDMA type of service</td>
<td>RDMA type of service</td>
<td>connRDMATypeOfService</td>
</tr>
<tr>
<td>Tcp only filter file</td>
<td>Path to a file with a list of no RDMA IP ranges</td>
<td>connTcpOnlyFilterFile</td>
</tr>
</tbody>
</table>

...continues
### 10.3 Managing The Deployed BeeGFS Instance

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to beegfs-client</th>
</tr>
</thead>
<tbody>
<tr>
<td>Log level</td>
<td>Log level</td>
<td>logLevel</td>
</tr>
<tr>
<td>Enable Quota</td>
<td>Enable quota</td>
<td>quotaEnabled</td>
</tr>
<tr>
<td>Create hardlinks as</td>
<td>Create a symlink when an application tries to create a hardlink</td>
<td>sysCreateHardlinksAsSymlinks</td>
</tr>
<tr>
<td>symlinks</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mount sanity check</td>
<td>Time in ms that server has to respond after mount sanity check</td>
<td>sysMountSanityCheckMS</td>
</tr>
<tr>
<td>ms</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Session check on close</td>
<td>Check for valid sessions on storage server when a file is closed</td>
<td>sysSessionCheckOnClose</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Sync on close</td>
<td>Sync file content on close</td>
<td>sysSyncOnClose</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Target offline timeout</td>
<td>Timeout until all storage targets are considered offline when no target</td>
<td>sysTargetOfflineTimeoutSecs</td>
</tr>
<tr>
<td></td>
<td>state updates can be fetched from management server</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Update target states</td>
<td>Interval for storage targets states check</td>
<td>sysUpdateTargetStatesSecs</td>
</tr>
<tr>
<td>time</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Enable xattrs</td>
<td>Enable xattrs</td>
<td>sysXAttrsEnabled</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Enable ACLs</td>
<td>Enable ACLs</td>
<td>sysACLsEnabled</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>File cache type</td>
<td>File read/write cache type</td>
<td>tuneFileCacheType</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Preferred meta file</td>
<td>Path to a file with preferred metadata servers</td>
<td>tunePreferredMetaFile</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Preferred storage file</td>
<td>Path to a file with preferred storage targets</td>
<td>tunePreferredStorageFile</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Remote fsync</td>
<td>Should fsync be executed on server to flush cached file</td>
<td>tuneRemoteFSync</td>
</tr>
</tbody>
</table>

© Bright Computing, Inc.
...continued

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to beegfs-client</th>
</tr>
</thead>
<tbody>
<tr>
<td>Use global append</td>
<td>Should files, opened in append mode, be protected by locks on local machine</td>
<td>tuneUseGlobalAppendLocks</td>
</tr>
<tr>
<td>locks</td>
<td>(false) or on servers (true)</td>
<td></td>
</tr>
<tr>
<td>Use global file</td>
<td>Should advisory locks be checked on local machine (false) or on servers (true)</td>
<td>tuneUseGlobalFileLocks</td>
</tr>
<tr>
<td>locks</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 10.5: BeeGFSClient role parameters and beegfs-client options

The beegfs-client settings are updated by Bright Cluster Manager in /etc/beegfs/beegfs-client.conf.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Option to beegfs-client</th>
</tr>
</thead>
<tbody>
<tr>
<td>Authentication</td>
<td>Path to the shared secret authentication file</td>
<td>connAuthFile</td>
</tr>
<tr>
<td>file</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Helper port TCP</td>
<td>TCP port for helper service</td>
<td>connHelperdPortTCP</td>
</tr>
<tr>
<td>No log date</td>
<td>Do not show date along with time in log</td>
<td>logNoDate</td>
</tr>
<tr>
<td>Number of log lines</td>
<td>Number of lines in log file, after which it is rotated</td>
<td>logNumLines</td>
</tr>
<tr>
<td>Number of log</td>
<td>Number of old log files to keep</td>
<td>logNumRotatedFiles</td>
</tr>
<tr>
<td>rotated files</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Log file</td>
<td>Path to the log file, empty means logs go to the journal</td>
<td>logStdFile</td>
</tr>
<tr>
<td>Helper workers number</td>
<td>Number of worker threads for helper service</td>
<td>tuneNumWorkers</td>
</tr>
</tbody>
</table>

Table 10.6: BeeGFSClient role parameters and beegfs-helperd options

The beegfs-helperd settings are updated by Bright Cluster Manager within /etc/beegfs/beegfs-helperd.conf.

The settings can be managed in cmsh via the beegfs::client role:

Example

```
[bright82 ~]# cmsh
[bright82]% category roles default
[bright82->category[default]->roles]% assign bee<TAB><TAB><TAB> beegfs::admon beegfs::client beegfs::management beegfs::metadata beegfs::storage
```

© Bright Computing, Inc.
10.3 Managing The Deployed BeeGFS Instance

```
[beegfs::client]
assign beegfs::client
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add services</td>
<td>yes</td>
</tr>
<tr>
<td>Name</td>
<td>BeeGFS::Client</td>
</tr>
<tr>
<td>Provisioning associations</td>
<td>&lt;0 internally used&gt;</td>
</tr>
<tr>
<td>Revision</td>
<td></td>
</tr>
<tr>
<td>Management node</td>
<td></td>
</tr>
<tr>
<td>Authentication file</td>
<td></td>
</tr>
<tr>
<td>Client port UDP</td>
<td>8004</td>
</tr>
<tr>
<td>Communication retry time</td>
<td>10m</td>
</tr>
<tr>
<td>Fallback expiration time</td>
<td>15m</td>
</tr>
<tr>
<td>Helper port TCP</td>
<td>8006</td>
</tr>
<tr>
<td>Interfaces file</td>
<td></td>
</tr>
<tr>
<td>Max internode number</td>
<td>12</td>
</tr>
<tr>
<td>Mountpoint</td>
<td>/mnt/beegfs</td>
</tr>
<tr>
<td>Net filter file</td>
<td></td>
</tr>
<tr>
<td>RDMA buffer size</td>
<td>8192</td>
</tr>
<tr>
<td>RDMA buffers number</td>
<td>70</td>
</tr>
<tr>
<td>RDMA type of service</td>
<td>0</td>
</tr>
<tr>
<td>Tcp only filter file</td>
<td></td>
</tr>
<tr>
<td>Use RDMA</td>
<td>yes</td>
</tr>
<tr>
<td>Log file</td>
<td></td>
</tr>
<tr>
<td>Log level</td>
<td>3</td>
</tr>
<tr>
<td>No log date</td>
<td>no</td>
</tr>
<tr>
<td>Number of log lines</td>
<td>50000</td>
</tr>
<tr>
<td>Number of log rotated files</td>
<td>5</td>
</tr>
<tr>
<td>Enable Quota</td>
<td>no</td>
</tr>
<tr>
<td>Create hardlinks as symlinks</td>
<td>no</td>
</tr>
<tr>
<td>Enable ACLs</td>
<td>no</td>
</tr>
<tr>
<td>Enable xattrs</td>
<td>no</td>
</tr>
<tr>
<td>Mount sanity check ms</td>
<td>11s</td>
</tr>
<tr>
<td>Session check on close</td>
<td>no</td>
</tr>
<tr>
<td>Sync on close</td>
<td>no</td>
</tr>
<tr>
<td>Target offline timeout</td>
<td>15m</td>
</tr>
<tr>
<td>Update target states time</td>
<td>1m</td>
</tr>
<tr>
<td>File cache type</td>
<td>buffered</td>
</tr>
<tr>
<td>Helper workers number</td>
<td>2</td>
</tr>
<tr>
<td>Preferred meta file</td>
<td></td>
</tr>
<tr>
<td>Preferred storage file</td>
<td></td>
</tr>
<tr>
<td>Remote fsync</td>
<td>yes</td>
</tr>
<tr>
<td>Use global append locks</td>
<td>no</td>
</tr>
<tr>
<td>Use global file locks</td>
<td>no</td>
</tr>
</tbody>
</table>

10.3.3 Usage

Accessing The Filesystem

After the installation process finishes, the BeeGFS filesystem can be accessed on the client nodes or on the head node. The default mount point is `/mnt/beegfs`, which can be changed within `/etc/beegfs/beegfs-mounts.conf`.

10.3.4 Admon Interface

After Admon is installed, then the Admon GUI jar file can be downloaded from the Admon node `<admon-node>`, via the Admon port `<admon-http-port>`, using a URL of the form (figure 10.17):

© Bright Computing, Inc.
When the administrator runs the client, configuration options are requested (figure 10.18). The hostname of the Admon node should be specified:

```
Hostname where the daemon runs: 10.2.61.102
HTTP Port the daemon uses: 8000
Internal desktop resolution: 2048x2048
Log level: 3
```

There are two users by default:

- **Information**, with the default password `information`. This user can only view BeeGFS statistics.
- **Administrator**, with the default password `admin`. This user can carry out BeeGFS administration.

It is highly recommended to change the default passwords after the first login.

After a successful login, the Admon interface becomes available for use (figure 10.20):
10.3 Managing The Deployed BeeGFS Instance

Post-Installation Software Management

Some time after Bright Cluster Manager has been installed, administrators may wish to manage other software on the cluster. This means carrying out software management actions such as installation, removal, updating, version checking, and so on.

Since Bright Cluster Manager is built on top of an existing Linux distribution, it is best that the administrator use distribution-specific package utilities for software management.

Packages managed by the distribution are hosted by distribution repositories. SUSE and RHEL distributions require the purchase of their license in order to access their repositories. The other distributions do not.

Packages managed by Bright Cluster Manager are hosted by the Bright Computing repository. Access to the Bright Computing repositories also requires a license (Chapter 4 of the Installation Manual). Available packages for a particular Bright Cluster Manager version and distribution can be viewed via the package dashboard at https://support.brightcomputing.com/packages-dashboard/.

There may also be software that the administrator would like to install that is outside the default packages collection. These could be source files that need compilation, or packages in other repositories.

A software image (section 2.1.2) is the filesystem that a node picks up from a provisioner (a head node or a provisioning node) during provisioning. A subtopic of software management on a cluster is software image management—the management of software on a software image. By default, a node uses the same distribution as the head node for its base image along with necessary minimal, cluster-mandated changes. A node may however deviate from the default, and be customized by having software added to it in several ways.

This chapter covers the techniques of software management for the cluster.

Section 11.1 describes the naming convention for a Bright Cluster Manager RPM or .deb package.
Section 11.2 describes how an RPM or .deb package is managed for the head node.
Section 11.3 describes how an RPM or .deb kernel package can be managed on a head node or image.
Section 11.4 describes how an RPM or .deb package can be managed on the software image.
Section 11.5 describes how a software other than an RPM or .deb package can be managed on a software image.
Section 11.6 describes how custom software images are created that are completely independent of the existing software image distribution and version.

11.1 Bright Cluster Manager Packages And Their Naming Convention

Like the distributions it runs on top of, Bright Cluster Manager uses either .rpm packages, managed by RPM (RPM Package Manager), or .deb (Debian) packages, managed by APT (Advanced Package Tool). For example, the cmdaemon package built by Bright Cluster Manager has the following .rpm and .deb packages:
The file name has the following structure:

```plaintext
package-version-revision_cm.x_y_hash.architecture.rpm
```

and

```plaintext
package_version-revision_cm.x_y-hash.architecture.deb
```

where:

- `package (cmdaemon)` is the name of the package
- `version (8.2)` is the version number of the package
- `revision (137541)` is the revision number of the package
- `cm` is used to indicate it is a package built by Bright Computing for the cluster manager
- `x.y (8.2)` is the version of Bright Cluster Manager for which the RPM was built
- `hash (075522ab7d or f315823e4a)` is a hash, and is only present for Bright Cluster Manager packages. It is used for reference by the developers of Bright Cluster Manager.
- `architecture (x86_64 for RPMs or amd64 for APT)` is the architecture for which the package was built. The architecture name of x86_64 or amd64 refers the same 64-bit x86 physical hardware in either case.

The differences in .rpm versus .deb package names are just some underbar/hyphen (_/-) changes, the hash (only for Bright Cluster Manager packages), and the architecture naming convention.

Among the distributions supported by Bright Cluster Manager, only Ubuntu uses .deb packages. The rest of the distributions use .rpm packages.

**Querying The Packages**

To check whether Bright Computing or the distribution has provided a file that is already installed on the system, the package it has come from can be found.

**For RPM-based systems:** `rpm -qf` can be used with the full path of the file:

**Example**

```bash
[root@bright82 ~]# rpm -qf /usr/bin/zless
gzip-1.5.-8.el7.x86_64
[root@bright82 ~]# rpm -qf /cm/local/apps/cmd/sbin/cmd
cmdaemon-8.2-127642_cm8.2.2d2f835e895.x86_64
```

In the example, `/usr/bin/zless` is supplied by the distribution, while `/cm/local/apps/cmd/sbin/cmd` is supplied by Bright Computing, as indicated by the "_cm" in the nomenclature.

**For APT-based systems:** A similar check can be done using `dpkg -S` to find the .deb package that provided the file, and then `dpkg -s` on the package name to reveal further information:

**Example**
11.2 Managing Packages On The Head Node

As an aside, system administrators should be aware that the Bright Computing version of a package is provided and used instead of a distribution-provided version for various technical reasons. The most important one is that it is tested and supported by Bright Cluster Manager. Replacing the Bright Computing version with a distribution-provided version can result in subtle and hard-to-trace problems in the cluster, and Bright Computing cannot offer support for a cluster that is in such a state, although some guidance may be given in special cases.

More information about the RPM Package Manager is available at http://www.rpm.org, while APT is documented for Ubuntu at http://manpages.ubuntu.com/manpages/.

11.2 Managing Packages On The Head Node

11.2.1 Managing RPM Or .deb Packages On The Head Node

Once Bright Cluster Manager has been installed, distribution packages and Bright Cluster Manager software packages are conveniently managed using the yum, zypper or apt repository and package managers. The zypper tool is recommended for use with the SUSE distribution, the apt utility is recommended for use with Ubuntu, and yum is recommended for use with the other distributions that Bright Cluster Manager supports. YUM is not set up by default in SUSE, and it is better not to install and use it with SUSE unless the administrator is familiar with configuring YUM.

Listing Packages On The Head Node With YUM and Zypper

For YUM and zypper, the following commands list all available packages:

```
yum list
```

or

```
zypper refresh; zypper packages
```

For zypper, the short command option `pa` can also be used instead of `packages`.

Listing Packages On The Head Node With APT

For Ubuntu, the `apt-cache` command is used to view available packages. To generate the cache used by the command, the command:

```
apt-cache gencaches
```

can be run.

A verbose list of available packages can then be seen by running:

```
apt-cache dumpavail
```
It is usually more useful to use the search option to apt-cache to search for the package with a regex:

```
apt-cache search <regex>
```

A similar, but slightly more verbose option is the search option for apt:

```
apt search <regex>
```

**Updating/Installing Packages On The Head Node**

To install a new package called `<package name>` into a distribution, the corresponding package managers are used as follows:

```
yum install <package name>
zypper in <package name>       # for SLES
apt install <package name>     # for Ubuntu
```

Installed packages can be updated to the latest by the corresponding package manager as follows:

```
yum update
zypper refresh; zypper up       # refresh recommended to update package metadata
apt update; apt upgrade         # update recommended to update package metadata
```

**An aside on the differences between the update, refresh/up, and update/upgrade options of the package managers:** The update option in YUM by default installs any new packages. On the other hand, the refresh option in zypper, and the update option in APT only update the meta-data (the repository indices). Only if the meta-data is up-to-date will an update via zypper, or an upgrade via apt install any newly-known packages. For convenience, in the Bright Cluster Manager manuals, the term update is used in the YUM sense in general—that is, to mean including the installation of new packages—unless otherwise stated.

Bright Computing maintains YUM and zypper repositories of its packages at:

```
http://updates.brightcomputing.com/yum
```

and updates are fetched by YUM and zypper for Bright Cluster Manager packages from there by default, to overwrite older package versions by default.

For Ubuntu, the Bright Computing .deb package repositories are at:

```
http://updates.brightcomputing.com/deb
```

Accessing the repositories manually (i.e. not using yum, zypper, or apt) requires a username and password. Authentication credentials are provided upon request. For more information on this, supportteam@brightcomputing.com should be contacted.

**Cleaning Package Caches On The Head Node**

The repository managers use caches to speed up their operations. Occasionally these caches may need flushing to clean up the index files associated with the repository. This can be done by the appropriate package manager with:

```
yum clean all
zypper clean -a       # for SUSE
apt-get clean         # for Ubuntu
```

© Bright Computing, Inc.
Signed Package Verification
As an extra protection to prevent Bright Cluster Manager installations from receiving malicious up-
dates, all Bright Cluster Manager packages are signed with the Bright Computing GPG public key
(0x5D849C16), installed by default in /etc/pki/rpm-gpg/RPM-GPG-KEY-cm for Red Hat, Scientific Linux,
and CentOS packages. The Bright Computing public key is also listed in Appendix B.
The first time YUM or zypper are used to install updates, the user is asked whether the Bright Compu-
ting public key should be imported into the local repository packages database. Before answering
with a “Y”, yum users may choose to compare the contents of /etc/pki/rpm-gpg/RPM-GPG-KEY-cm with
the key listed in Appendix B to verify its integrity. Alternatively, the key may be imported into the local
RPM database directly, using the following command:

```
rpm --import /etc/pki/rpm-gpg/RPM-GPG-KEY-cm
```

With APT the Bright Cluster Manager keyring is already imported into /etc/apt/trusted.gpg.d/
brightcomputing-archive-cm.gpg if the cm-config-apt package, provided by the Bright Computing
repository, has been installed. The cm-config-apt package is installed by default for the Ubuntu edition
of Bright Cluster Manager.

Third Party Packages
Installation of the following third party packages, most of which are repackaged by Bright Computing
for installation purposes, is described in Chapter 7 of the Installation Manual. These are installed mostly
on the head node, but some packages work as a server and client process, with the clients installed and
running on the regular nodes:

Most of the third party packages in the following list are repackaged by Bright Computing for instal-
lation purposes. These are installed mostly on the head node, but some packages work as a server and
client process, with the clients installed and running on the regular nodes. The packages are described
in Chapter 7 of the Installation Manual:

- Modules (section 7.1)
- Shorewall (section 7.2)
- Compilers (section 7.3):
  - GCC (section 7.3.1)
  - Intel Compiler Suite (section 7.3.2)
  - PGI High-Performance Compilers (section 7.3.3)
  - FLEXlm License Daemon (section 7.3.4)
- CUDA (section 7.4)
- Lustre (section 7.8)

Exclusion of packages on the head node can be carried out as explained in section 11.3.2, where the
kernel package is used as an example for exclusion.

Gnome And KDM Installation—Disabling network-manager
NetworkManager interferes with the custom network management of Bright Cluster Manager and should
thus never be enabled.

When installing the cluster from bare metal using the Bright Cluster Manager installer, if the X11
graphical user installation option checkbox is ticked, then NetworkManager is disabled by using a finalize
script, so that Bright Cluster Manager’s networking is able to work properly.

However, some time after the cluster is configured, the administrator may wish to install the Gnome
or KDE graphical desktop environment. This can be done in YUM or APT. For example in RHEL the
installation can be carried out with:
yum groupinstall "GNOME Desktop"

The Network Manager package is a dependency of the Gnome and KDE desktops, and is therefore automatically installed by the package manager command. As a result, the NetworkManager service is enabled, and the distribution defaults are to have it run. However, because this interferes with the custom network management of Bright Cluster Manager, the NetworkManager should be disabled.

Disabling Network Manager can be done as follows:

- On RHEL 6 and derivatives:

  Example

  ```
  [root@bright82 ~]# service NetworkManager status
  NetworkManager (pid 1527) is running...
  [root@bright82 ~]# chkconfig NetworkManager off
  [root@bright82 ~]# service NetworkManager stop
  ```

- On RHEL 7 and derivatives, and Ubuntu:

  Example

  ```
  [root@bright82 ~]# systemctl status NetworkManager.service
  ...
  [root@bright82 ~]# systemctl disable NetworkManager.service
  ```

- On SLES-based systems:

  Example

  YAST can be used to disable NetworkManager by going to Network Devices → Network Settings → Global Options → Network Setup Method, and setting the option: Traditional Method with ifup.

11.2.2 Installation Of Packages On The Head Node That Are Not .deb And Not .rpm Packages

Sometimes a package is not packaged as an RPM or .deb package by Bright Computing or by the distribution. In that case, the software can usually be treated as for installation onto a standard distribution. There may be special considerations on placement of components that the administrator may feel appropriate due to the particulars of a cluster configuration.

For example, for compilation and installation of the software, some consideration may be made of the options available on where to install parts of the software within the default shared filesystem. A software may have a compile option, say --prefix, that places an application <application> in a directory specified by the administrator. If the administrator decides that <application> should be placed in the shared directory, so that everyone can access it, the option could then be specified as: "--prefix=/cm/shared/apps/<application>".

Other commonly provided components of software for the applications that are placed in shared may be documentation, licenses, and examples. These may similarly be placed in the directories /cm/shared/docs, /cm/shared/licenses, and /cm/shared/examples. The placement may be done with a compiler option, or, if that is not done or not possible, it could be done by modifying the placement by hand later. It is not obligatory to do the change of placement, but it helps with cluster administration to stay consistent as packages are added.

Module files (section 2.2 of this manual, and 7.1 of the Installation Manual) may sometimes be provided by the software, or created by the administrator to make the application work for users easily
with the right components. The directory `/cm/shared/modulefiles` is recommended for module files to do with such software.

To summarize the above considerations on where to place software components, the directories under `/cm/shared` that can be used for these components are:

```
/cm/shared/
|-- apps
|-- docs
|-- examples
|-- licenses
|-- modulefiles
```

### 11.3 Kernel Management On A Head Node Or Image

Care should be taken when updating a head node or software image. This is particularly true when custom kernel modules compiled against a particular kernel version are being used.

A package can be managed in a software image and the image deployed to nodes. A careful administrator typically clones a copy of a working image that is known to work, before modifying the image.

#### 11.3.1 Installing A Standard Distribution Kernel Into An Image Or On A Head Node

A standard distribution kernel is treated almost like any other package in a distribution.

This means that:

- For head nodes, installing a standard kernel is done according to the normal procedures of managing a package on a head node (section 11.2).

- For regular nodes, installing a standard distribution kernel is done according to the normal procedures of managing an package inside an image, via a changed root (chroot) directory (section 11.4), but with some special aspects that are discussed in this section.

When a kernel is updated or reinstalled (section 11.3.3), kernel-specific drivers, such as OFED drivers may need to be updated or reinstalled. OFED driver installation details are given in section 7.6 of the *Installation Manual*.

#### Kernel Package Name Formats

For RHEL, individual kernel package names take a form such as:

```
kernel-3.10.0-327.3.1.el7.x86_64.rpm
```

The actual one suited to a cluster varies according to the distribution used. RPM Packages with names that begin with "kernel-devel-“ are development packages that can be used to compile custom kernels, and are not required when installing standard distribution kernels.

For Ubuntu, individual Linux kernel image package names take a form such as:

```
linux-image-*.deb
```

or

```
linux-signed-image-*.deb
```

Running `apt-cache search ˆlinux | grep 'kernel image'` shows the various packaged kernel images in the distribution.

#### Other Extra Considerations

When installing a kernel, besides the chroot steps of section 11.4, extra considerations for kernel packages are:

- The kernel must also be explicitly set in CMDaemon (section 11.3.3) before it may be used by the regular nodes.
• Some GRUB-related errors with a text such as "grubby fatal error: unable to find a suitable template" typically show up during the installation of a kernel package in the software image. These occur due to a failure to find the partitions when running the post-install scripts in the chroot environment. They can simply be ignored because the nodes do not boot from partitions configured by GRUB.

• Warnings such as: warning: Failed to read auxiliary vector, /proc not mounted? may come up, due to /proc not being in use in the software image. These can also simply be ignored.

• The ramdisk of a regular node must be regenerated using the createramdisk command (section 11.4.3).

• If the cluster is in a high availability configuration, then installing a new kernel on to the active head node may in some cases stop its network interface, and trigger a failover. It is therefore usually wiser to make the change on the passive head node first, or to disable automatic failover, before carrying out a change that could initiate a failover.

As is standard for Linux, both head or regular nodes must be rebooted to use the new kernel.

11.3.2 Excluding Kernels And Other Packages From Updates

Specifying A Kernel Or Other Package For Update Exclusion

Sometimes it may be desirable to exclude the kernel from updates on the head node.

• When using yum, to prevent an automatic update of a package, the package is listed after using the --exclude flag. So, to exclude the kernel from the list of packages that should be updated, the following command can be used:

   ```
   yum --exclude kernel update
   ```

   To exclude a package such as kernel permanently from all YUM updates, without having to specify it on the command line each time, the package can instead be excluded inside the repository configuration file. YUM repository configuration files are located in the /etc/yum.repos.d directory, and the packages to be excluded are specified with a space-separated format like this:

   ```
   exclude = <package 1> <package 2> ...
   ```

• The zypper command can also carry out the task of excluding the kernel package from getting updated when updating. To do this, the kernel package is first locked (prevented from change) using the addlock command, and the update command is run. Optionally, the kernel package is unlocked again using the removelock command:

   ```
   zypper addlock kernel
   zypper update
   zypper removelock kernel       #optional
   ```

• One APT way to upgrade the software while excluding the kernel image package is to first update the system, then to mark the kernel as a package that is to be held, and then to upgrade the system. Optionally, after the upgrade, the hold mark can be removed:

   ```
   apt update
   apt-mark hold <linux-image-version>
   apt upgrade
   apt-mark unhold <linux-image-version>     #optional
   ```

   The complementary way to carry out an upgrade in APT while holding the kernel back, is to use pinning. Pinning can be used to set dependency priorities during upgrades. Once set, it can hold a particular package back while the rest of the system upgrades.

© Bright Computing, Inc.
### Specifying A Repository For Update Exclusion

Sometimes it is useful to exclude an entire repository from an update on the head node. For example, the administrator may wish to exclude updates to the parent distribution, and only want updates for the cluster manager to be pulled in. In that case, in RHEL-derivatives a construction such as the following may be used to specify that only the repository IDs matching the glob `cm*` are used, from the repositories in `/etc/yum.repos.d/`:

```
[root@bright82 ~]# yum repolist
...
repo id  repo name      status
base     CentOS-6 - Base  6,356+11
cm-rhel6-8.2 Cluster Manager 8.2  316+13
cm-rhel6-8.2-updates Cluster Manager 8.2  514
extras   CentOS-6 - Extras  14
updates  CentOS-6 - Updates 391
repolist: 7,591
[root@bright82 ~]# yum --disablerepo=* --enablerepo=cm* update
```

In Ubuntu, repositories can be added or removed by editing the repository sources under `/etc/apt/sources.list.d/`. There is also the `apt edit-sources` command, which, unsurprisingly, also edits the repository sources. The `add-apt-repository` command (`man (1) add-apt-repository`) edits the repository sources by line. Running `add-apt-repository -h` shows options and examples.

#### 11.3.3 Updating A Kernel In A Software Image

A kernel is typically updated in the software image by carrying out a package installation using the `chroot` environment (section 11.4), or specifying a relative root directory setting.

Package dependencies can sometimes prevent the package manager from carrying out the update, for example in the case of OFED packages (section 7.6 of the *Installation Manual*). In such cases, the administrator can specify how the dependency should be resolved.

Parent distributions are by default configured, by the distribution itself, so that only up to 3 kernel images are kept when installing a new kernel with the package manager. However, in a Bright Cluster Manager cluster, this default distribution value is overridden by a default Bright Cluster Manager value, so that kernel images are never removed during YUM updates, or `apt upgrade`, by default.

For a software image, if the kernel is updated by the package manager, then the kernel is not used on reboot until it is explicitly enabled with either Bright View or `cmsh`.

- To enable it using Bright View, the Kernel version entry for the software image should be set. This can be accessed via the clickpath Provisioning→Software images→Edit→Settings→Kernel version (figure 11.1).
436 Post-Installation Software Management

Figure 11.1: Updating A Software Image Kernel With Bright View

- To enable the updated kernel from cmsh, the softwareimage mode is used. The kernelversion property of a specified software image is then set and committed:

Example

```
[root@bright82 ~]# cmsh
[bright82]% softwareimage
[bright82]->softwareimage% use default-image
[bright82->softwareimage[default-image]]% set kernelversion 3.10.0-327.3.1.el7.x86_64
[bright82->softwareimage*[default-image*]]% commit -w
```

Tab-completion suggestions for the `set kernelversion` command will display the available values for the kernel version.

11.3.4 Setting Kernel Options For Software Images

A standard kernel can be booted with special options that alter its functionality. For example, a kernel can boot with `apm=off`, to disable Advanced Power Management, which is sometimes useful as a workaround for nodes with a buggy BIOS that may crash occasionally when it remains enabled.

In Bright View, to enable booting with this kernel option setting, the clickpath Provisioning→Software images→Edit→Settings→Kernel parameters (figure 11.1) is used to set the kernel parameter to `apm=off` for that particular image.

In cmsh, the equivalent method is to modify the value of "kernel parameters" in softwareimage mode for the selected image:

Example

```
[root@bright82 ~]# cmsh
[bright82]% softwareimage
[bright82]->softwareimage% use default-image
[bright82->softwareimage[default-image]]% append kernelparameters " apm=off"
[bright82->softwareimage*[default-image*]]% commit
```

© Bright Computing, Inc.
Often kernel options load up modules and their parameters. Making module loading persist after reboot and setting module loading order is covered in section 5.3.2.

Some kernel options may require changes to be made in the BIOS settings in order to function.

### 11.3.5 Kernel Driver Modules

Bright Computing provides some packages which install new kernel drivers or update kernel drivers. In RPM-based distributions, such packages generally require the `kernel-devel` package. In this section, the `kernel-devel-check` utility is first described, followed by the various drivers that Bright Computing provides.

#### Kernel Driver Modules: `kernel-devel-check` Compilation Check

For RPM, the distribution’s `kernel-devel` package is required to compile kernel drivers for its kernel. It must be the same version and release as the kernel running on the node. For APT, the `linux-header` package corresponding to the kernel image is used.

In RPM-based distributions, to check the head node and software images for the installation status of the `kernel-devel` package, the Bright Cluster Manager utility `kernel-devel-check` is run from the head node:

**Example**

```
[root@mycluster ~]# kernel-devel-check
Head node: mycluster
  Found kernel development rpm package kernel-devel-3.10.0-327.3.1.el7.x86_64

Software image: default-image
  No kernel development directories found, probably no kernel development package installed.
  Kernel development rpm package kernel-devel-3.10.0-327.3.1.el7.x86_64 not found
  If needed, try to install the kernel development package with:
  # chroot /cm/images/default-image yum install kernel-devel-3.10.0-327.3.1.el7.x86_64

Software image: default-image1
  No kernel development directories found, probably no kernel development package installed.
  Kernel development rpm package kernel-devel-3.10.0-327.3.1.el7.x86_64 not found
  If needed, try to install the kernel development package with:
  # chroot /cm/images/default-image1 yum install kernel-devel-3.10.0-327.3.1.el7.x86_64

As suggested by the output of `kernel-devel-check`, running a command on the head node such as:

```
[root@mycluster ~]# chroot /cm/images/default-image1 yum install kernel-devel-3.10.0-327.3.1.el7.x86_64
```

installs a `kernel-devel` package, to the software image called `default-image1` in this case. The package version suggested corresponds to the kernel version set for the image, rather than necessarily the latest one that the distribution provides.

#### Kernel Driver Modules: Improved Intel Wired Ethernet Drivers

**Improved Intel wired Ethernet drivers—what they are:** The standard distributions provide Intel wired Ethernet driver modules as part of the kernel they provide. Bright Computing provides an improved version of the drivers with its own `intel-wired-ethernet-drivers` package. The package contains more recent versions of the Intel wired Ethernet kernel drivers: `e1000`, `e1000e`, `igb`, `igbvf`, `ixgbe` and `ixgbevf`. They often work better than standard distribution modules when it comes to performance, features, or stability.
Improved Intel wired Ethernet drivers—replacement mechanism:  The improved drivers can be installed on all nodes.

For head nodes, the standard Intel wired Ethernet driver modules on the hard drive are overwritten by the improved versions during package installation. Backing up the standard driver modules before installation is recommended, because it may be that some particular hardware configurations are unable to cope with the changes, in which case reverting to the standard drivers may be needed.

For regular nodes, the standard distribution wired Ethernet drivers are not overwritten into the provisioner’s software image during installation of the improved drivers package. Instead, the standard driver modules are removed from the kernel and the improved modules are loaded to the kernel during the init stage of boot.

For regular nodes in this “unwritten” state, removing the improved drivers package from the software image restores the state of the regular node, so that subsequent boots end up with a kernel running the standard distribution drivers from on the image once again. This is useful because it allows a very close-to-standard distribution to be maintained on the nodes, thus allowing better distribution support to be provided for the nodes.

If the software running on a fully-booted regular node is copied over to the software image, for example using the “Grab to image” button (section 11.5.2), this will write the improved driver module into the software image. Restoring to the standard version is then no longer possible with simply removing the improved drivers packages. This makes the image less close-to-standard, and distribution support is then less easily obtained for the node.

Thus, after the installation of the package is done on a head or regular node, for every boot from the next boot onwards, the standard distribution Intel wired Ethernet drivers are replaced by the improved versions for fully-booted kernels. This replacement occurs before the network and network services start. The head node simply boots from its drive with the new drivers, while a regular node initially starts with the kernel using the driver on the software image, but then if the driver differs from the improved one, the driver is unloaded and the improved one is compiled and loaded.

Improved Intel wired Ethernet drivers—installation:  The drivers are compiled on the fly on the regular nodes, so a check should first be done that the kernel-devel package is installed on the regular nodes (section 11.3.5).

If the regular nodes have the kernel-devel package installed, then the following yum commands are issued on the head node, to install the package on the head node and in the default-image:

**Example**

```
[root@mycluster ~]# yum install intel-wired-ethernet-drivers
[root@mycluster ~]# chroot /cm/images/default-image
[root@mycluster /]# yum install intel-wired-ethernet-drivers
```

For SUSE, the equivalent zypper commands are used (“zypper in” instead of “yum install”).

Kernel Driver Modules: CUDA Driver Installation

CUDA drivers are drivers the kernel uses to manage GPUs. These are compiled on the fly for nodes with GPUs in Bright Cluster Manager. The details of how this is done is covered in the CUDA software section (section 7.4 of the Installation Manual).

Kernel Driver Modules: OFED And OPA Stack Installation

By default, the distribution provides the OFED stack used by the kernel to manage the InfiniBand or RDMA interconnect. Installing a Bright Cluster Manager repository OFED stack to replace the distribution version is covered in section 7.6 of the Installation Manual. Some guidance on placement into initrd for the purpose of optional InfiniBand-based node provisioning is given in section 5.3.3.

Installing a Bright Cluster Manager repository Omni-Path (OPA) stack is covered in section 7.7 of the Installation Manual.
11.4 Managing A Package In A Software Image And Running It On Nodes

A package can be managed in a software image and the image deployed to nodes. A careful administrator typically clones a copy of a working image that is known to work, before modifying the image.

11.4.1 Installing From Head Into The Image: Changing The Root Directory Into Which The Packages Are Deployed

Managing packages (including the kernel) inside a software image is most easily done while on the head node, using a “change root” mechanism with `rpm`, `yum`, `zypper`, or `apt`. The change root mechanism can be used as a package manager option, or can be invoked as a standalone command.

**Change Root As An Option In The Package Manager Command**

**Using the rpm command:** The `rpm` command supports the `--root` flag. To install an RPM package inside the default software image while in the head node environment, using the repositories of the head node, the command can be used as follows:

**Example**

```
rpm --root /cm/images/default-image -ivh /tmp/libxml2-2.6.16-6.x86_64.rpm
```

**Using the yum command:** The `yum` command allows more general updates with a change root option. For example, all packages in the default image can be updated using `yum` for RHEL and derivatives with:

**Example**

```
yum --installroot=/cm/images/default-image update #for RHEL variants
```

A useful option to restrict the version to which an image is updated, is to use the option `--releasever`. For example, to allow only updates up to RHEL7.4, the command in the preceding example would have `--releasever=7.4` appended to it.

**Using the zypper command:** For SLES, `zypper` can be used as follows to update the image:

**Example**

```
zypper --root /cm/images/default-image up #for SLES
```

**Change Root With chroot, Then Running The Package Manager Commands**

If the repositories used by the software image are the same as the repositories used by the head node, then the `chroot` command can be used instead of the `--installroot`/`--root` options to get the same result as the package manager options. That is, the same result is accomplished by first chrooting into an image, and subsequently executing the `rpm`, `yum`, or `zypper` commands without `--root` or `--installroot` arguments. Thus:

**For RHEL and derivatives:** For YUM-based update, running `yum update` is recommended to update the image, after using the `chroot` command to reach the root of the image:

**Example**

```
[root@bright82 ~]# chroot /cm/images/default-image
[root@bright82 /]# yum update #for RHEL variants
...updates happen...
[root@bright82 /]# exit #get out of chroot
```

© Bright Computing, Inc.
For SLES: For SLES, running `zypper up` is recommended to update the image, after using the `chroot` command to reach the root of the image:

**Example**

```
bright82:~# chroot /cm/images/default-image
bright82.cm.cluster:/ # zypper up #for SLES
... updates happen...
bright82.cm.cluster:/ # exit #get out of chroot
```

For Ubuntu: For Ubuntu and APT, the software image often requires having the `/proc`, `/sys`, and `/dev` directories available during the package installation in the chroot jail. Additionally, the `/proc` namespace used should not be that of the head node due to namespace issues that affect decision-making in some of the pre- and post-installation script bundled with the package.

Pre-configuring all this with bind mounting before going into the chrooted filesystem is a little tedious. Therefore a Bright Cluster Manager utility, `cm-chroot-sw-image`, can be used to take care of this.

Thus, for Ubuntu, if the cluster administrator would like to run `apt update; apt upgrade` to update the image, then the recommended way to do it is to start the process with the `cm-chroot-sw-image` command:

**Example**

```
root@bright82:~# cm-chroot-sw-image /cm/images/default-image
... messages indicate that the special directories have been mounted automatically, and a chroot jail has been entered...
root@bright82:/# apt update; apt upgrade #for Ubuntu
... An upgrade session runs in the image root. Some administrator inputs may be needed...
root@bright82:/# exit #get out of chroot
... messages indicate that the special directories have been unmounted automatically...
```

Excluding Packages And Repositories From The Image

Sometimes it may be desirable to exclude a package or a repository from an image.

- If using `yum --installroot`, then to prevent an automatic update of a package, the package is listed after using the `--exclude` flag. For example, to exclude the kernel from the list of packages that should be updated, the following command can be used:

  ```
yum --installroot=/cm/images/default-image --exclude kernel update
  ```

  To exclude a package such as `kernel` permanently from all YUM updates, without having to specify it on the command line each time, the package can instead be excluded inside the repository configuration file of the image. YUM repository configuration files are located in the `/cm/images/default-image/etc/yum.repos.d` directory, and the packages to be excluded are specified with a space-separated format like this:

  ```
  exclude = <package 1> <package 2> ...
  ```

- The `zypper` command can also carry out the task of excluding a package from getting updated when during update. To do this, the package is first locked (prevented from change) using the `addlock` command, then the update command is run, and finally the package is unlocked again using the `removelock` command. For example, for the `kernel` package:

  ```
  zypper --root /cm/images/default-image addlock kernel
  zypper --root /cm/images/default-image update
  zypper --root /cm/images/default-image removelock kernel
  ```
• For Ubuntu, the `apt-mark hold` command can be used to exclude a package. This is described in the particular case of excluding the kernel package earlier on, in section 11.3.2.

• Sometimes it is useful to exclude an entire repository from an update to the image. For example, the administrator may wish to exclude updates to the base distribution, and only want Bright Cluster Manager updates to be pulled into the image. In that case, a construction like the following may be used to specify that, for example, from the repositories listed in `/cm/images/default-image/etc/yum.repos.d/`, only the repositories matching the pattern `cm*` are used:

```
[root@bright82 ~]# cd /cm/images/default-image/etc/yum.repos.d/
[root@bright82 yum.repos.d]# yum --installroot=/cm/images/default-image/etc/yum.repos.d/ --disablerepo=* --enablerepo=cm* update
```

• For Ubuntu, excluding a repository can be carried out by removing the repository under `/etc/apt/sources.list.d/`. Slightly handier may be to use the `add-apt-repository` command, or the `apt edit-sources` command.

### 11.4.2 Installing From Head Into The Image: Updating The Node

If the images are in place, then the node that use those images do not run those images until they have the changes placed on the nodes. Rebooting the nodes that use the software images is a straightforward way to have those nodes start up with the new images. Alternatively, the nodes can usually simply be updated without a reboot (section 5.6), if no reboot is required by the underlying Linux distribution.

### 11.4.3 Installing From Head Into The Image: Possible Issues When Using rpm --root, yum --installroot Or chroot

• The update process on an image, when using YUM, zypper, or APT, will fail to start if the image is being provisioned by a provisioner at the time. The administrator can either wait for provisioning requests to finish, or can ensure no provisioning happens by setting the image to a locked state (section 5.4.7), before running the update process. The image can then be updated. The administrator normally unlocks the image after the update, to allow image maintenance by the provisioners again.

**Example**

```
[root@bright82 ~]# cmsh -c "softwareimage use default-image; set locked yes; commit"
[root@bright82 ~]# yum --installroot /cm/images/default-image update
[root@bright82 ~]# cmsh -c "softwareimage use default-image; set locked no; commit"
```

• The `rpm --root` or `yum --installroot` command can fail if the versions between the head node and the version in the software image differ significantly. For example, installation from a Scientific Linux 6 head node to a Red Hat 7 software image is not possible with those commands, and can only be carried out with chroot.

• While installing software into a software image with an `rpm --root`, `yum --installroot` or with a chroot method is convenient, there can be issues if daemons start up in the image, or if the distribution installation scripts exit with errors due to being in an image environment rather than a real instance.

For example, installation scripts that stop and re-start a system service during a package installation may successfully start that service within the image’s chroot jail and thereby cause related, unexpected changes in the image. Pre- and post- (un)install scriptlets that are part of RPM or APT packages may cause similar problems.

Bright Computing’s RPM and .deb packages are designed to install under chroot without issues. However packages from other repositories may cause the issues described. To deal with that, the
cluster manager runs the chroot process health check, which alerts the administrator if there is a daemon process running in the image. The chroot process also checks and kills the process if it is a crond process.

- For some package updates, the distribution package management system attempts to modify the ramdisk image. This is true for kernel updates, many kernel module updates, and some other packages. Such a modification is designed to work on a normal machine. For a regular node on a cluster, which uses an extended ramdisk, the attempt does nothing.
In such cases, a new ramdisk image must nonetheless be generated for the regular nodes, or the nodes will fail during the ramdisk loading stage during start-up (section 5.8.4).

The ramdisk image for the regular nodes can be regenerated manually, using the createramdisk command (section 5.3.2).

- Trying to work out what is in the image from under chroot must be done with some care.
For example, under chroot, running “uname -a” returns the kernel that is currently running—that is the kernel outside the chroot. This is typically not the same as the kernel that will load on the node from the filesystem under chroot. It is the kernel in the filesystem under chroot that an unwary administrator may wrongly expect to detect on running the uname command under chroot.

To find the kernel version that is to load from the image, the software image kernel version property (section 11.3.3) can be inspected using the cluster manager with:

```
Example

cmsh -c "softwareimage; use default-image; get kernelversion"
```

### 11.5 Managing Non-RPM Software In A Software Image And Running It On Nodes

Sometimes, packaged software is not available for a software image, but non-packaged software is. This section describes the installation of non-packaged software onto a software image in these two cases:

1. copying only the software over to the software image (section 11.5.1)
2. placing the software onto the node directly, configuring it until it is working as required, and syncing that back to the software image using Bright Cluster Manager’s special utilities (section 11.5.2)

In both cases, before making changes, a careful administrator typically clones a copy of a working image that is known to work, before modifying the image.

As a somewhat related aside, completely overhauling the software image, including changing the base files that distinguish the distribution and version of the image is also possible. How to manage that kind of extreme change is covered separately in section 11.6.

However, this current section (11.5) is about modifying the software image with non-RPM software while staying within the framework of an existing distribution and version.

In all cases of installing software to a software image, it is recommended that software components be placed under appropriate directories under /cm/shared (which is actually outside the software image).

So, just as in the case for installing software to the head node in section 11.2.2, appropriate software components go under:

```
/cm/shared/
|-- apps
|-- docs
|-- examples
|-- licenses
`-- modulefiles
```

© Bright Computing, Inc.
11.5 Managing Non-RPM Software In A Software Image And Running It On Nodes

11.5.1 Managing The Software Directly On An Image

The administrator may choose to manage the non-packaged software directly in the correct location on the image.

For example, the administrator may wish to install a particular software to all nodes. If the software has already been prepared elsewhere and is known to work on the nodes without problems, such as for example library dependency or path problems, then the required files can simply be copied directly into the right places on the software image.

The chroot command may also be used to install non-packaged software into a software image. This is analogous to the chroot technique for installing packages in section 11.4:

Example

cd /cm/images/default-image/usr/src

tar -xvzf /tmp/app-4.5.6.tar.gz

chroot /cm/images/default-image

cd /usr/src/app-4.5.6

./configure --prefix=/usr

make install

Whatever method is used to install the software, after it is placed in the software image, the change can be implemented on all running nodes by running the updateprovisioners (section 5.2.4) and imageupdate (section 5.6.2) commands.

11.5.2 Managing The Software Directly On A Node, Then Syncing Node-To-Image

Why Sync Node-To-Image?

Sometimes, typically if the software to be managed is more complex and needs more care and testing than might be the case in section 11.5.1, the administrator manages it directly on a node itself, and then makes an updated image from the node after it is configured, to the provisioner.

For example, the administrator may wish to install and test an application from a node first before placing it in the image. Many files may be altered during installation in order to make the node work with the application. Eventually, when the node is in a satisfactory state, and possibly after removing any temporary installation-related files on the node, a new image can be created, or an existing image updated.

Administrators should be aware that until the new image is saved, the node loses its alterations and reverts back to the old image on reboot.

The node-to-image sync can be seen as the converse of the image-to-node sync that is done using imageupdate (section 5.6.2).

The node-to-image sync discussed in this section is done using the “Grab to image” or “Synchronize image” menu option from Bright View, or using the “grabimage” command with appropriate options in cmsh. The sync automatically excludes network mounts and parallel filesystems such as Lustre and GPFS, but includes any regular disk mounted on the node itself.

Some words of advice and a warning are in order here:

- The cleanest, and recommended way, to change an image is to change it directly in the node image, typically via changes within a chroot environment (section 11.5.1).

- Changing the deployed image running on the node can lead to unwanted changes that are not obvious. While many unwanted changes are excluded because of the excludelistgrab* lists during a node-to-image sync, there is a chance that some unwanted changes do get captured. These changes can lead to unwanted or even buggy behavior. The changes from the original deployed image should therefore be scrutinized with care before using the new image.

- For scrutiny, the bash command:
vimdiff <(cd image1; find . | sort) <(cd image2; find . | sort)

run from /cm/images/ shows the changed files for image directories image1 and image2, with uninteresting parts folded away. The <(commands) construction is called process substitution, for administrators unfamiliar with this somewhat obscure technique.

**Node-To-Image Sync Using Bright View**

In Bright View, saving the node state from, for example, node001 to a new image is done by selecting the appropriate menu option from the clickpath Devices → Nodes[node001] → Edit → Settings → Actions → Software image → option (figure 11.2).

The possible options for saving a node state to an image are:

1. The **Grab to image** option: This opens up a dialog offering an image to sync to. Selecting the image creates a new image, grabbing what is to go to the image from the node. It wipes out whatever (if anything) is in the selected image, except for a list of excluded items. The excluded items are specified in the “Exclude list grabbing to a new image” list, available under Node Categories via the clickpath Grouping → Node categories[default-image] → Settings → Exclude list grab new. The exclude list is known as excludelistgrabnew (page 446) in cmsh.

2. The **Synchronize image** option: This does a sync from the node back to the software image that the node is provisioned with, using evaluation based on file change detection between the node and the image. It is thus a synchronization to the already existing software image that is currently in use by the node.

The items that it excludes from the synchronization are specified in the “Exclude list image grab” list, available under Node Categories via the clickpath Grouping → Node categories[default-image] → Settings → Exclude list grab. This exclude list is known as excludelistgrab (page 446) in cmsh.
The synchronize option can be viewed by the administrator as being a more “gentle” way to carry out a node-to-image sync, compared to the grab option—that is, a “gentle sync” that tries to avoid wiping out existing files, versus a “violent grab” that can wipe out existing files.

The exclude lists are there to ensure, among other things, that the configuration differences between nodes are left alone for different nodes with the same image. The exclude lists are simple by default, but they conform in structure and patterns syntax in the same way that the exclude lists detailed in section 5.4.7 do, and can therefore be quite powerful.

The images that are available for selection with the Grab to image option can be existing images, while the image that the Synchronize image option syncs to is the existing image. If such existing images are known to work well with nodes, then overwriting them with a new image on a production system may be reckless. A wise administrator who has prepared a node that is to write an image would therefore follow a process similar to the following instead of simply overwriting an existing image:

1. A new image can be cloned from the old image via the clickpath Provisioning→Software images→Clone, and setting a name for the new image, for example: newimage. The node state with the software installed on it would then be saved using the Grab to image option, and choosing the image name newimage as the image to save it to.

2. A new category is then cloned from the old category via the clickpath Grouping→Categories→Clone, and setting a name for the new category, for example newcategory. The old image in newcategory is changed to the new image newimage via the clickpath Grouping→Categories→Edit→Settings→Software image→newimage.

3. A newly-cloned category has no nodes initially. Some nodes are set to the new category so that their behavior with the new image can be tested. The chosen nodes can be made members of the new category from within the Settings option of each node, and saving the change. The clickpath for this is Devices→Nodes→Edit→Settings→Category→newcategory

4. The nodes that have been placed in the new category are now made to pick up and run their new images. This can be done with a reboot of those nodes.

5. After sufficient testing, all the remaining nodes can be moved to using the new image. The old image is removed if no longer needed, or perhaps kept around just in case for reference.

**Node-To-Image Sync Using cmsh**

The preceding Bright View method can alternatively be carried out using cmsh commands. The cmsh equivalent to the Synchronize image option is the grabimage command, available from device mode. The cmsh equivalent to the Grab to image option is the grabimage -i command, where the -i option specifies the new image it will write to. As before, that image must be created or cloned beforehand.

Cloning an image for use, setting a category of nodes that will use it, and then synchronizing a node that has the new software setup over to the new image on the provisioner might be carried out as follows via cmsh:

```
[root@bright82 ~]# cmsh
[bright82]% softwareimage
[bright82->softwareimage]% clone default-image default-image1
[bright82->softwareimage*][default-image1]% commit
[bright82->softwareimage][default-image1]% category
[bright82->category]% clone default default1
[bright82->category*][default1]% commit
[bright82->category][default1]% set softwareimage default-image1
[bright82->category*][default1]% commit
[bright82->category][default1]% device
[bright82->device]% grabimage -w -i default-image1 node001
[bright82->device]%
```
The **grabimage** command without the `-w` option simply does a dry-run so that the user can see in the provisioning logs what should be grabbed, without having the changes actually carried out. Running **grabimage -w** instructs CMDaemon to really write the image.

When writing out the image, two exclude lists may be used:

- The `excludelistgrabnew` object. This is used with **grabimage** with the `-i` option. The list can be accessed and edited under **cmsh**, in its category mode for a node image, as the `excludelistgrabnew` object. It corresponds to the “Exclude list grabbing to a new image” exclusion list associated with the **Grab to image option** (page 444) in Bright View.

- The `excludelistgrab` object. This is used with the **grabimage** command, run without the `-i` option. The list can be accessed and edited under **cmsh**, in its category mode for a node image, as the `excludelistgrab` object. It corresponds to the “Exclude list image grab” exclusion list associated with the **Synchronize image option** (page 444) in Bright View.

### 11.6 Creating A Custom Software Image

By default, the software image used to boot non-head nodes is based on the same version and release of the Linux distribution as used by the head node. However, sometimes an image based on a different distribution or a different release from that on the head node may be needed.

A custom software image is created typically by building an entire filesystem image from a regular node. The node, which is never a head node, is then called the **base host**, with the term “base” used to indicate that it has no additional cluster manager packages installed. The distribution on the base host, is called the **base distribution** and is a selection of packages derived from the **parent distribution** (Red Hat, Scientific Linux etc). A **base distribution package** is a package or rpm that is directly provided by the vendor of the parent distribution which the base distribution is based on, and is not provided by Bright Cluster Manager.

Creating a custom software image consists of two steps. The first step (section 11.6.1) is to create a **base (distribution) archive** from an installed base host. The second step (section 11.6.2) is to create the image from the base archive using a special utility, **cm-create-image**.

#### 11.6.1 Creating A Base Distribution Archive From A Base Host

**Structure Of The Base Distribution Archive**

The step of creating the base distribution archive is done by creating an archive structure containing the files that are needed by the non-head node.

The filesystem that is archived in this way can differ from the special way that a Linux distribution unpacks and installs its filesystem on to a machine. This is because the distribution installer often carries out extra changes, for example in GRUB boot configuration. The creation of the base distribution archive is therefore a convenience to avoid working with the special logic of a distribution installer, which will vary across distributions and versions. Instead, the filesystem and contents of a node on which this parent distribution is installed—i.e. the end product of that logic—is what is dealt with.

The archive can be a convenient and standard **tar.gz** file archive (sometimes called the “base tar”), or, taking the step a little further towards the end result, the archive can be a fully expanded archive file tree.

**Repository Access Considerations When Intending To Build A Base Distribution Archive**

For convenience, the archive should be up-to-date. So, the base host used to generate the base distribution archive should ideally have updated files. If, as is usual, the base host is a regular node, then it should ideally be up to date with the repositories that it uses. Therefore running **yum update** or **zypper**
up on the base host image, and then provisioning the image to the base host, is recommended in order to allow the creation of an up-to-date base distribution archive.

However sometimes updates are not possible or desirable for the base host. This means that the base host archive that is put together from the base host filesystem is an un-updated archive. The custom image that is to be created from the archive must then be also be created without accessing the repositories, in order to avoid dependency problems with the package versions. Exclusion of access to the repositories is possible by specifying options to the `cm-create-image` command, and is described in section 11.6.2.

**Examples Of How To Build A Base Distribution Archive**

In the following example, a base distribution `tar.gz` archive `/tmp/BASEDIST.tar.gz` is created from the base host `basehost64`:

**Example**

```bash
ssh root@basehost64 "
  *tar -cz --acls --xattrs \
  --exclude /etc/HOSTNAME --exclude /etc/localtime \n  --exclude /proc --exclude /lost+found --exclude /sys \n  --exclude /root/.ssh --exclude /var/lib/dhcpcd/* \n  --exclude /media/floppy --exclude /etc/motd \n  --exclude /root/.bash_history --exclude /root/CHANGES \n  --exclude /etc/udev/rules.d/*persistent*.rules \n  --exclude /var/spool/mail/* --exclude /rhn \n  --exclude /etc/sysconfig/rhn/systemid --exclude /tmp/* \n  --exclude /etc/sysconfig/rhn/systemid.save \n  --exclude /root/mbox --exclude /var/cache/yum/* \n  --exclude /etc/cron.daily/rhn-updates /" > /tmp/BASEDIST.tar.gz
```

Or alternatively, a fully expanded archive file tree can be created from `basehost64` by rsyncing to an existing directory (here it is `/cm/images/new-image`):

**Example**

```bash
rsync -av --acls --xattrs --hard-links --numeric-ids \
  --exclude=/etc/HOSTNAME --exclude=/etc/localtime --exclude=/proc \
  --exclude=/lost+found --exclude=/sys --exclude=/root/.ssh \n  --exclude=/var/lib/dhcpcd/* --exclude=/media/floppy \n  --exclude=/etc/motd --exclude=/root/.bash_history \n  --exclude=/CHANGES --exclude=/var/spool/mail/* \n  --exclude=/etc/udev/rules.d/*persistent*.rules \n  --exclude=/rhn --exclude=/etc/sysconfig/rhn/systemid \n  --exclude=/etc/sysconfig/rhn/systemid.save --exclude=/tmp/* \n  --exclude=/var/spool/up2date/* --exclude=/var/log/* \n  --exclude=/root/mbox --exclude=/var/cache/yum/* \n  --exclude=/etc/cron.daily/rhn-updates \
  root@basehost64:/ /cm/images/new-image/
```

The `--acls`, `--xattrs`, and `--hard-links` options are only explicitly required—if they are valid—for earlier distribution versions of RHEL and SLES and their derivatives. This is indicated by the following table:

© Bright Computing, Inc.
<table>
<thead>
<tr>
<th>Used by default?</th>
<th>--acls</th>
<th>--xattrs</th>
<th>--hard-links</th>
</tr>
</thead>
<tbody>
<tr>
<td>before RHEL7</td>
<td>no</td>
<td>no</td>
<td>no</td>
</tr>
<tr>
<td>before SLES12</td>
<td>*</td>
<td>*</td>
<td>no</td>
</tr>
<tr>
<td>RHEL7 and beyond</td>
<td>yes</td>
<td>yes</td>
<td>yes</td>
</tr>
<tr>
<td>SLES 12 and beyond</td>
<td>yes</td>
<td>yes</td>
<td>yes</td>
</tr>
</tbody>
</table>

* The --acls --xattrs options in both examples are invalid before SLES12, so can be dropped.

The defaults can be modified by adjusting the AdvancedConfig options RsyncHardLinks (page 653), RsyncXattrs (page 653), and RsyncAcls (page 653).

For distribution versions RHEL7 and its derivatives, and also for SLES12, extended attributes and ACLs are used as rsync options by default. This is expected to continue for future distribution versions.

To use SELinux on compute nodes for RHEL7 and its derivatives, RsyncXattrs must be explicitly disabled (that is, extended attributes must not be copied).

For versions of RHEL and derivatives prior to RHEL7, the --acls and --xattrs options should be specified explicitly. For versions of SLES prior to SLES12, the --acls and --xattrs options are invalid, so can be dropped.

Having built the archive by following the examples suggested, the first step in creating the software image is now complete.

### 11.6.2 Creating The Software Image With cm-create-image

The second step, that of creating the image from the base archive, now needs to be done. This uses the cm-create-image utility, which is part of the cluster-tools package.

The cm-create-image utility uses the base archive as the base for creating the image. By default, it expects that the base distribution repositories be accessible just in case files need to be fetched from a repository package.

Thus, when the cm-create-image utility is run with no options, the image created mostly picks up the software only from the base archive. However, the image picks up software from the repository packages:

- if it is required as part of a dependency, or
- if it is specified as part of the package selection file (page 452).

If a repository package file is used, then it should be noted that the repository package files may be more recent compared with the files in the base archive. This can result in an image with files that are perhaps unexpectedly more recent in version than what might be expected from the base archive, which may cause compatibility issues. To prevent this situation, the --exclude option (section 11.2) can be used to exclude updates for the packages that are not to be updated.

Repository access can be directly to the online repositories provided by the distribution, or it can be to a local copy. For RHEL, online repository access can be activated by registering with the Red Hat Network (section 5.1 of the Installation Manual). Similarly, for SUSE, online repository access can be activated by registering with Novell (section 5.2 of the Installation Manual). An offline repository can be constructed as described in section 11.6.3 of this manual.

### Usage Of The cm-create-image Command

The usage information for cm-create-image lists options and examples:

**Usage:**

```bash
cm-create-image <OPTIONS1> [OPTIONS2]
```

**Options1:**
11.6 Creating A Custom Software Image

---------
-a | --fromarchive <archive> Create software image from archive file of supported base distribution. Supported file formats are .tar, .tgz, .tar.gz, .tbz, and .tar.bz2. The extension must match the format.

-d | --fromdir <dir path> Create software image from existing directory that already has valid base distribution.

-h | --fromhost <hostname> Create software image from running host

-n | --imagename <name> Name of software image to create in cluster management daemon database.

OPTIONS2:
---------
-i | --imagedir <dir name> Name of directory to be created in /cm/images. Contents of archive file are extracted into this directory (default: name specified with -n).

-r | --recreate Recreate directory specified by -i or default, if it exists.

-s | --skipdist Skip install of base distribution packages

-e | --excludemrepo Do not copy default cluster manager repo files. (Use this option when the repo files have been already modified in the image directory, and hence must not be overwritten.)

-f | --forcecreate Force non-interactive mode

-u | --updateimage If image specified by -n already exists, then it is updated, with the new parameters

-b | --basedistrepo <file> Use this as the repo configuration file for fetching required distribution packages (cannot be used with -e).

-c | --cmrepo <file> Use this as the repo configuration file for fetching required cluster manager packages (cannot be used with -e).

-m | --minimal Install only a minimal set of packages, relevant for the cluster management daemon to function. Use with -s option, to also prevent additional distribution packages from being installed.

-w | --hwvendor Install hardware vendor specific packages. Valid choices are: dell|cray|ciscoucs|hp|ibm|supermicro|other

-l | --resolvconf resolv.conf to be used inside image dir, during image creation (by default /etc/resolv.conf from head node is used)

-x | --excludemcm <list> List of CM packages to exclude (comma-separated)

-o | --exclude-from <file> File containing the exclude patterns in the format of the 'rsync' command. See the 'exclude/include pattern rules' in the documentation of 'rsync' for more information.

© Bright Computing, Inc.
-j | --excludedist <list>  List of distribution packages to exclude
                          (comma-separated)
-q | --excludehwvendor <list> List of hardware vendor packages to
                            exclude (comma-separated)
-g | --enableextrarepo <value> Argument must be the special string
                            'public' or path to a directory that has
                            the Bright DVD/ISO mounted.

EXAMPLES:
---------
1. cm-create-image -a /tmp/RHEL6.tar.gz -n rhel6-image
2. cm-create-image -a /tmp/RHEL6.tar.gz -n rhel6-image -i /cm/images/test-image
3. cm-create-image -d /cm/images/SLES11-image -n sles11-image
4. cm-create-image -h node001 -n node001-image
5. cm-create-image -a /tmp/RHEL6.tar.gz -n rhel6-image -i /cm/images/new-image -r
6. cm-create-image -a /tmp/RHEL6.tar.gz -n rhel6-image -i /cm/images/new-image -u
7. cm-create-image -d /cm/images/new-image -n bio-image -s -e
8. cm-create-image -d /cm/images/new-image -n bio-image -s -b /tmp/rhel6-updates.repo
9. cm-create-image -d /cm/images/new-image -n bio-image -s -c /tmp/cm-rhel6.repo
10. cm-create-image -d /cm/images/new-image -n bio-image -s -m

Explanations Of The Examples In Usage Of cm-create-image
Explanations of the 10 examples in the usage text follow:

1. In the following, a base distribution archive file, /tmp/RHEL6.tar.gz, is written out to a software
   image named rhel6-image:

   cm-create-image --fromarchive /tmp/RHEL6.tar.gz --imagename rhel6-image

   The image with the name rhel6-image is created in the CMDaemon database, making it available
   for use by cmsh and Bright View. If an image with the above name already exists, then
   /cm/create-image will exit and advise the administrator to provide an alternate name.
   By default, the image name specified sets the directory into which the software image is installed.
   Thus here the directory is /cm/images/rhel6-image/.

2. Instead of the image getting written into the default directory as in the previous item, an alternative
directory can be specified with the --imagedir option. Thus, in the following, the base
   distribution archive file, /tmp/RHEL6.tar.gz is written out to the /cm/images/test-image directory.
   The software image is given the name rhel6-image:

   cm-create-image --fromarchive /tmp/RHEL6.tar.gz --imagename rhel6-image --imagedir \
   /cm/images/test-image

3. If the contents of the base distribution file tree have been transferred to a directory, then no extraction
   is needed. The --fromdir option can then be used with that directory. Thus, in the following, the archive
   has already been transferred to the directory /cm/images/SLES11-image, and it is that directory which is then used to place the image under a directory named
   /cm/images/sles11-image/. Also, the software image is given the name sles11-image:

   cm-create-image --fromdir /cm/images/SLES11-image --imagename sles11-image

4. A software image can be created from a running node using the --fromhost option. This option
   makes cm-create-image behave in a similar manner to grabimage (section 11.5.2) in cmsh. It requires
   passwordless access to the node in order to work. Generic nodes, that is nodes outside
   the cluster, can also be used. An image named node001-image can then be created from a running
   node named node001 as follows:
11.6 Creating A Custom Software Image

```
cm-create-image --fromhost node001 --imagename node001-image
```

By default the image goes under the `/cm/images/node001-image/` directory.

5. If the destination directory already exists, the `--recreate` option can be used to recreate the existing directory. The administrator should be aware that this means removal of the content of any existing directory of the same name. Thus, in the following, the content under `/cm/images/new-image/` is deleted, and new image content is taken from the base distribution archive file, `/tmp/RHEL6.tar.gz` and then placed under `/cm/images/new-image/`. Also, the software image is given the name `rhel6-image`:

```
cm-create-image --fromarchive /tmp/RHEL6.tar.gz --imagename rhel6-image --imagedir /cm/images/new-image --recreate
```

If the `--recreate` option is not used, then the contents are simply overwritten, that is, the existing directory contents are copied over by the source content. It also means that old files on the destination directly may survive unchanged because the new source may not have filenames matching those.

6. The destination directory can also just be updated without removing the existing contents, by using the option `--updateimage`. The option is almost the same as the "contents are simply overwritten" behavior described in example 5, but it actually works like an `rsync` command. Thus, in the following, the base distribution archive file, `/tmp/RHEL6.tar.gz`, is used to update the contents under the directory `/cm/images/new-image/`. The name of the image is also set to `rhel6-image`:

```
cm-create-image --fromarchive /tmp/RHEL6.tar.gz --imagename rhel6-image --imagedir /cm/images/new-image --updateimage
```

7. With the default Bright Cluster Manager, the head node provisions a software image based on the parent distribution to the other nodes. The software image which runs on the nodes provides a selection of distribution packages from the parent distribution.

The default software image is thus a selection of Red Hat packages, if the head node uses Red Hat, or a selection of SUSE packages if the head node uses SUSE, and so on. The other packages for the software image are supplied by Bright Computing.

When creating a custom software image, and if using the `--skipdist` flag with `cm-create-image`, then Bright Cluster Manager packages are added to the software image, but no parent distribution packages are added. Thus in the following, the packages made available to `cm-create-image` in the directory `/cm/images/new-image/`, are installed into the image named `bio-image`; however, no packages matching parent distribution packages are installed (because of the `--skipdist` option). Furthermore, transfer of the packages takes place only if they are newer than the files already in the `bio-image` image (because of the `--updateimage` option):

```
cm-create-image --fromdir /cm/images/new-image --imagename bio-image --skipdist --updateimage
```

So, only Bright Cluster Manager packages are updated to the image `bio-image` in the directory `/cm/images/bio-image`.

8. The `--basedistrepo` flag is used together with a `.repo` file. The file defines the base distribution repository for the image. The file is copied over into the repository directory of the image,(`/etc/yum.repos.d/` for Red Hat and similar, or `/etc/zypp/repos.d/` for SLES).

9. The `--cmrepo` flag is used together with a `.repo` file. The file defines the cluster manager repository for the image. The file is copied over into the repository directory of the image,(`/etc/yum.repos.d/` for Red Hat and similar, or `/etc/zypp/repos.d/` for SLES).
10. The --minimal flag tells cm-create-image to install only a limited number of Bright packages and its dependencies, which are required for the cluster management daemon to run the node. This also means that no Bright configuration RPMS will be installed, and all existing system configuration files will remain untouched. The minimal package selection list is read from /cm/local/apps/cluster-tools/config/minimal. The word “minimal” primarily refers to Bright Cluster Manager, and it is still required to use the --skipdist option explained earlier, in order to prevent additional parent distribution packages from being installed. This is because “minimal” in the context of the parent distribution can vary a lot depending on the requirements of different users, and is beyond the scope of cm-create-image.

```
cm-create-image --fromdir /cm/images/new-image --imagename bio-image --skipdist --minimal
```

**Package Selection Files In cm-create-image**

Regarding explanation 7 in the preceding explanations text, the selection of packages on the head node is done using a package selection file.

Package selection files are available in /cm/local/apps/cluster-tools/config/. For example, if the base distribution of the software image being created is CentOS6, then the configuration file used is:

```
/cm/local/apps/cluster-tools/config/CENTOS6-config-dist.xml
```

The package selection file is made up of a list of XML elements, specifying the name of the package, architecture and image type. For example:

```
...<package image="slave" name="apr" arch="x86_64"/>
 <package image="slave" name="apr-util" arch="x86_64"/>
 <package image="slave" name="atk-devel" arch="x86_64"/>
 <package image="slave" name="autoconf" arch="noarch"/>
...
```

The minimal set of packages in the list defines the minimal distribution that works with Bright Cluster Manager, and is the base-distribution set of packages, which may not work with some features of the distribution or Bright Cluster Manager. To this minimal set the following packages may be added to create the custom image:

- Packages from the standard repository of the parent distribution. These can be added to enhance the custom image or to resolve a dependency of Bright Cluster Manager. For example, in the (parent) Red Hat distribution, packages can be added from the (standard) main Red Hat channel to the base-distribution.

- Packages from outside the standard repository, but still from inside the parent distribution. These can be added to enhance the custom image or to resolve a dependency of Bright Cluster Manager. For example, outside the main Red Hat channel, but still within the parent distribution, there is a supplementary packages channel (Red Hat 5) or an optional packages channel (Red Hat 6). Packages from these optional/supplementary channels can be added to the base-distribution to enhance the capabilities of the image or resolve dependencies of Bright Cluster Manager. Section 7.4.1 of the *Installation Manual* considers an example of such a dependency for the CUDA package.

Unless the required distribution packages and dependencies are installed and configured, particular features of Bright Cluster Manager, such as CUDA, cannot work correctly or cannot work at all.

The package selection file also contains entries for the packages that can be installed on the head (image="master") node. Therefore non-head node packages must have the image="slave" attribute.
11.6 Creating A Custom Software Image

Kernel Module Selection By `cm-create-image`

For an image created by `cm-create-image`, with a distribution `<dist>`, the default list of kernel modules to be loaded during boot are read from the file `/cm/local/apps/cluster-tools/config/<dist>-slavekernelmodules`.

`<dist>` can take the value CENTOS6, CENTOS6u10, CENTOS7, CENTOS7u5, RHEL6, RHEL6u10, RHEL7, RHEL7u5, SL6, SL6u10, SL7, SL7u5, SLES12, SLES12sp3, SLES12sp4, SLES15, SLES15sp1.

If custom kernel modules are to be added to the image, they can be added to this file.

Output And Logging During A `cm-create-image` Run

The `cm-create-image` run goes through several stages: validation, sanity checks, finalizing the base distribution, copying Bright Cluster Manager repository files, installing distribution package, finalizing image services, and installing Bright Cluster Manager packages. An indication is given if any of these stages fail.

Further detail is available in the logs of the `cm-create-image` run, which are kept in `/var/log/cmcreateimage.log`. `<image name>`, where `<image name>` is the name of the built image.

Default Image Location

The default-image is at `/cm/images/default-image`, so the image directory can simply be kept as `/cm/images/`.

During a `cm-create-image` run, the `--imagedir` option allows an image directory for the image to be specified. This must exist before the option is used.

More generally, the full path for each image can be set:

- Using Bright View via the clickpath Provisioning→Software images→Software Images→Edit→Settings→Path
- In `cmsh` within softwareimage mode, for example:

  ```bash
  [bright82->softwareimage]% set new-image path /cm/higgs/new-images
  ```

- At the system level, the images or image directory can be symlinked to other locations for organizational convenience

Workload Manager Reconfiguration On The Custom Regular Node Image

After a custom regular node image has been created by `cm-create-image`, and the workload manager on the custom node image is to be managed by Bright Cluster Manager, then the manager typically needs to be reconfigured. This can be done by running the `wlm-setup` utility using the `--image` option with the path to the custom node image (section 7.3).

11.6.3 Configuring Local Repositories For Linux Distributions, And For The Bright Cluster Manager Package Repository, For A Software Image

Using local instead of remote repositories can be useful in the following cases:

- for clusters that have restricted or no internet access.
- for the RHEL and SUSE Linux distributions, which are based on a subscription and support model, and therefore do not have free access to their repositories.
- for creating a custom image with the `cm-create-image` command introduced in section 11.6.2, using local base distribution repositories.

The administrator can choose to access an online repository provided by the distribution itself via a subscription as described in Chapter 5 of the *Installation Manual*. Another way to set up a repository is
to set it up as a local repository, which may be offline, or perhaps set up as a locally-controlled proxy with occasional, restricted, updates from the distribution repository.

In the three procedures that follow, the first two procedures explain how to create and configure a local offline SLES zypper or RHEL YUM repository for the subscription-based base distribution packages. These first two procedures assume that the corresponding ISO/DVD has been purchased/downloaded from the appropriate vendors. The third procedure then explains how to create a local offline YUM repository from the Bright Cluster Manager ISO for CentOS so that a cluster that is completely offline still has a complete and consistent repository access.

Thus, a summary list of what these procedures are about is:

- Setting up a local repository for SLES (page 454)
- Setting up a local repository for RHEL (page 454)
- Setting up a local repository for CentOS and Bright from the Bright Cluster Manager ISO for CentOS (page 455)

**Configuring Local Repositories For SLES For A Software Image**

For SLES11 SP0, SLES11 SP1, and SLES11 SP2, the required packages are spread across two DVDs, and hence two repositories must be created. Assuming the image directory is `/cm/images/sles11sp1-image`, while the names of the DVDs are `SLES-11-SP1-SDK-DVD-x86_64-GM-DVD1.iso` and `SLES-11-SP1-DVD-x86_64-GM-DVD1.iso`, then the contents of the DVDs can be copied as follows:

```bash
mkdir /mnt1 /mnt2
mkdir /cm/images/sles11sp1-image/root/repo1
mkdir /cm/images/sles11sp1-image/root/repo2
mount -o loop,ro SLES-11-SP1-SDK-DVD-x86_64-GM-DVD1.iso /mnt1
cp -ar /mnt1/* /cm/images/sles11sp1-image/root/repo1/
mount -o loop,ro SLES-11-SP1-DVD-x86_64-GM-DVD1.iso /mnt2
cp -ar /mnt2/* /cm/images/sles11sp1-image/root/repo2/
```

The two repositories can be added for use by zypper in the image, as follows:

```bash
chroot /cm/images/sles11sp1-image
zypper addrepo /root/repo1 "SLES11SP1-SDK"
zypper addrepo /root/repo2 "SLES11SP1"
exit (chroot)
```

**Configuring Local Repositories For RHEL For A Software Image**

For RHEL distributions, the procedure is almost the same. The required packages are contained in one DVD.

```bash
mkdir /mnt1
mkdir /cm/images/rhel-image/root/repo1
mount -o loop,ro RHEL-DVD1.iso /mnt1
cp -ar /mnt1/* /cm/images/rhel-image/root/repo1/
```

The repository is added to YUM in the image, by creating the repository file `/cm/images/rhel-image/etc/yum.repos.d/rhel-base.repo` with the following contents:

```bash
[base]
name=Red Hat Enterprise Linux $releasever - $basearch - Base
baseurl=file:///root/repo1/Server
gpgcheck=0
enabled=1
```

© Bright Computing, Inc.
Configuring Local Repositories For CentOS And Bright Computing For A Software Image

Mounting the ISOs  The variable $imagedir is assigned as a shortcut for the software image that is to be configured to use a local repository:

```
imagedir=/cm/images/default-image
```

If the ISO is called bright-centos.iso, then its filesystem can be mounted by the root user on a new mount, /mnt1, as follows:

```
mkdir /mnt1
mount -o loop bright-centos.iso /mnt1
```

The head node can then access the ISO filesystem.

The same mounted filesystem can also be mounted with the bind option into the software image. This can be done inside the software image by the root user, in the same relative position as for the head node, as follows:

```
mkdir $imagedir/mnt1
mount -o bind /mnt1 $imagedir/mnt1
```

This allows an operation run under the $imagedir in a chroot environment to access the ISO filesystem too.

Creating YUM repository configuration files:  YUM repository configuration files can be created:

- **for the head node**: A repository configuration file

  `/etc/yum.repos.d/cm8.2-dvd.repo`

  can be created, for example, for a release tagged with a `<subminor>` number tag, with the content:

  ```
  [bright-repo]
  name=Bright Cluster Manager DVD Repo
  baseurl=file:///mnt1/data/cm-rpms/8.2-<subminor>
  enabled=1
  gpgcheck=1
  exclude = slurm* pbspro* sge* cm-hwloc
  ```

- **for the regular node image**: A repository configuration file

  `$imagedir/etc/yum.repos.d/cm6.1-dvd.repo`

  can be created. This file is in the image directory, but it has the same content as the previous head node repository configuration file.

Verifying that the repository files are set up right:  To verify the repositories are usable on the head node, the YUM cache can be cleaned, and the available repositories listed:

```
[root@bright82 ~]# yum clean all
[root@bright82 ~]# yum repolist -v
bright-repo Bright Cluster Manager DVD Repo
...
```

To carry out the same verification on the image, these commands can be run with `yum --installroot=$imagedir` substituted in place of just `yum`.

The ISO repository should show up, along with any others that are accessible. Connection attempts that fail to reach a network-based or local repositories display errors. If those repositories are not needed, they can be disabled from within their configuration files.
11.6.4 Creating A Custom Image From The Local Repository

After having created the local repositories for SLES, RHEL or CentOS (section 11.6.3), a custom software image based on one of these can be created. For example, for CentOS, in a directory given the arbitrary name offlineimage:

```
cm-create-image -d $imagedir -n offlineimage -e -s
```

The `-e` option prevents copying the default cluster manager repository files on top of the image being created, since they may have been changed by the administrator from their default status. The `-s` option prevents installing additional base distribution packages that might not be required.
Cluster Monitoring

Bright Cluster Manager monitoring allows a cluster administrator to monitor anything that can be monitored in the cluster. Much of the monitoring consists of pre-defined sampling configurations. If there is anything that is not configured, but the data on which it is based can be sampled, then monitoring can be configured for it too, by the administrator.

The monitoring data can be viewed historically, as well as on demand. The historical monitoring data can be stored raw, and optionally also as consolidated data—a way of summarizing data.

The data can be handled raw and processed externally, or it can be visualized within Bright View in the form of customizable charts. Visualization helps the administrator spot trends and abnormal behavior, and is helpful in providing summary reports for managers.

Monitoring can be configured to set off alerts based on triggers, and pre-defined or custom actions can be carried out automatically, depending on triggers. The triggers can be customized according to user-defined conditional expressions.

Carrying out such actions automatically after having set up triggers for them means that the monitoring system can free the administrator from having to carry out these chores.

In this chapter, the monitoring system is explained with the following approach:

1. A basic example is first presented in which processes are run on a node. These processes are monitored, and trigger an action when a threshold is exceeded.

2. With this easy-to-understand example as a basic model, the various features and associated functionality of the Bright Cluster Manager monitoring system are then described and discussed in further depth. These include visualization of data, concepts, configuration, monitoring customization and cmsh use.

12.1 A Basic Monitoring Example And Action

12.1.1 Synopsis Of Basic Monitoring Example

In section 12.1, after an overview (section 12.1.1), a minimal basic example of monitoring a process is set up (section 12.1.2) and used (section 12.1.3). The example is contrived, with the aim being to present a basic example that covers a part of what the monitoring system is capable of handling. The basic example gives the reader a structure to keep in mind, around which further details are fitted and filled in during the coverage in the rest of this chapter.

For the basic example, a user runs a large number of pointless CPU-intensive processes on a head node which is normally very lightly loaded. An administrator can monitor user mode CPU load usage throughout the cluster, and notices this usage spike. After getting the user to stop wasting CPU cycles, the administrator may decide that putting a stop to such processes automatically is a good idea. The administrator can set that up with an action that is triggered when a high load is detected. The action that is taken after triggering, is to stop the processes (figure 12.1).
High load detected and processes stopped

Figure 12.1: Monitoring Basic Example: CPU-intensive Processes Started, Detected And Stopped

The basic example thus illustrates how Bright Cluster Manager monitoring can be used to detect something on the cluster and how an action can be set up and triggered based on that detection.

12.1.2 Before Using The Basic Monitoring Example—Setting Up The Pieces

Running A Large Number Of Pointless CPU-Intensive Processes

One way to simulate a user running pointless CPU-intensive processes is to run several instances of the standard unix utility, yes. The yes command sends out an endless number of lines of “y” texts. It is typically used in scripts to answer dialog prompts for confirmation.

The administrator can run 8 subshell processes in the background from the command line on the head node, with yes output sent to /dev/null, as follows:

```bash
for i in {1..8}; do ( yes > /dev/null &); done
```

Running “mpstat 2” shows usage statistics for each processor, updating every 2 seconds. It shows that %user, which is user mode CPU usage percentage, is close to 90% on an 8-core or less head node when the 8 subshell processes are running.

Setting Up The Kill Action

To stop the pointless CPU-intensive yes processes, the command killall yes can be used. The administrator can make it a part of a script killallyes:

```bash
#!/bin/bash
killall yes
```

and make the script executable with a chmod 700 killallyes. For convenience, it may be placed in the /cm/local/apps/cmd/scripts/actions directory where some other action scripts also reside.
12.1 A Basic Monitoring Example And Action

12.1.3 Using The Monitoring Basic Example
Now that the pieces are in place, the administrator can use Bright View to add the killallyesaction action to its action list, and then set up a trigger for the action:

Adding The Action To The Actions List
In Bright View:

- The clickpath

  Monitoring Configuration → Actions → Monitoring Actions → killprocess → Clone

  is used to clone the structure of an existing action. The killprocess action is convenient because it is expected to function in a similar way, so its options should not have to be modified much. However, any action could be cloned and the clone modified in appropriate places.

- The name of the cloned action is changed. That is, the administrator sets Name to killallyesaction. This is just a sensible label—the name can be arbitrary.

- Script is set to the path /cm/local/apps/cmd/scripts/actions/killallyyes, which is where the script was placed earlier (page 458).

After saving, the killallyesaction action becomes part of the list of monitoring actions (figure 12.2).

![Figure 12.2: Bright View Monitoring Configuration: Adding An Action](image)

Setting Up A Trigger Using CPUUser On The Head Node(s)
The clickpath

  Monitoring Configuration → Triggers → Failing health checks → Clone

© Bright Computing, Inc.
can be used to configure a monitoring trigger, by cloning an existing trigger. A trigger is a sample state condition that runs an action. In this case, the sample state condition may be that the metric (section 12.2.3) \texttt{CPUUser} must not exceed 50. If it does, then an action (\texttt{killallyesaction}) is run, which should kill the \texttt{yes} processes.

- \texttt{CPUUser} is a measure of the time spent in user mode CPU usage per second, and is measured in jiffy intervals per second.

- A jiffy interval is a somewhat arbitrary time interval that is predefined for a kernel developers per platform. It is the minimum amount of time that a process has access to the CPU before the kernel can switch to another task.

Unlike \%\texttt{user} from the \texttt{top} command, a jiffy interval is not a percentage.

To configure this, the trigger attributes can be modified as follows (figure 12.3):

- A name is set for the trigger. The name can be arbitrary, and \texttt{killallyestrigger} is used in this example.

- the trigger is made active by enabling it

- the trigger can be set to run an action script if the sample state crosses over into a state that meets the trigger condition. That is, \texttt{Enter actions} is configured for a particular condition.

The condition under which the \texttt{Enter actions} action script is run in the example, can simply be when \texttt{CPUUser} on the head node is above 50. Such a condition can be set by setting an expression in the \texttt{expression} subwindow. In this subwindow (figure 12.4):
12.1 A Basic Monitoring Example And Action

Figure 12.4: Bright View Monitoring Configuration: Setting An Expression

- A name is set for the expression. The name can be arbitrary, and \texttt{killallyesregex} is used for \texttt{Name} in this example.
- An entity is set. In this case, the entity being monitored is the head node. If the head node is called \texttt{bright82} in this example, then \texttt{bright82} is the value set for \texttt{entities}. An entity is often simply a device, but it can be any object that CMDaemon stores.
- A measurable is set. In this case, \texttt{Measurables} is set to \texttt{CPUUser}.
- An operator and threshold value are set. In this case \texttt{GT}, which is the greater than operator, and \texttt{50} which is a significant amount of \texttt{CPUUser} time in jiffies/s, are set for \texttt{Operator} and \texttt{Value}.

After saving the configuration, the \texttt{killallyesregex} regular expression evaluates the data being sampled for the \texttt{killallyestrigger} trigger. If the expression is \texttt{TRUE}, then the trigger launches the \texttt{killallyesaction} action.

The Result

In the preceding section, an action was added, and a trigger was set up with a monitoring expression.

With a default installation on a newly installed cluster, the measurement of \texttt{CPUUser} is done every 120s (the period can be modified in the Data Producer window of Bright View, as seen in figure 12.9). The basic example configured with the defaults thus monitors if \texttt{CPUUser} on the head node has crossed the bound of 50 jiffies/s every 120s.

If \texttt{CPUUser} is found to have entered—that is crossed over from below the value and gone into the zone beyond 50 jiffies—then the \texttt{killallyesregex} regular expression notices that. Then, the trigger it is configured for, \texttt{killallyestrigger}, runs the \texttt{killallyesaction} action, which runs the \texttt{killallyes} script. The \texttt{killallyes} script kills all the running \texttt{yes} processes. Assuming the system is trivially loaded apart from these \texttt{yes} processes, the \texttt{CPUUser} metric value then drops to below 50 jiffies.

To clarify what “found to have entered” means in the previous paragraph:

After an \texttt{Enter} trigger condition has been met for a sample, the first sample immediately after that does not ever meet the \texttt{Enter} trigger condition, because an \texttt{Enter} threshold crossing condition requires the previous sample to be below the threshold.

The second sample can only launch an action if the \texttt{Enter} trigger condition is met and if the preceding sample is below the threshold.
Other non-yes CPU-intensive processes running on the head node can also trigger the killallyes script. Since the script only kills yes processes, leaving any non-yes processes alone, it would in such a case run unnecessarily. This is a deficiency due to the contrived and simple nature of the basic example which is being illustrated here. In a production case the action script is expected to have a more sophisticated design.

At this point, having gone through section 12.1, the reader is expected to have a rough idea of how monitoring, triggers, trigger conditional expressions, and actions work. The following sections in this chapter cover the concepts and features for Bright Cluster Manager monitoring in greater detail.

### 12.2 Monitoring Concepts And Definitions

A discussion of the concepts of monitoring, along with definitions of terms used, is appropriate at this point. The features of the monitoring system in Bright Cluster Manager covered later on in this chapter will then be understood more clearly.

#### 12.2.1 Measurables

Measurables are measurements (sample values) that are obtained via data producers (section 12.2.10) in CMDaemon’s monitoring system. The measurements can be made for nodes, head nodes, other devices, or other entities.

**Types Of Measurables**

Measurables can be:

- **enummetrics**: measurements with a small number of states. The states can be pre-defined, or user-defined. Further details on enummetrics are given in section 12.2.2.

- **metrics**: measurements with number values, and no data, as possible values. For example, values such as: -13113143234.5, 24, 9234131299. Further details on metrics are given in section 12.2.3.

- **health checks**: measurements with the states PASS, FAIL, and UNKNOWN as possible states, and no data as another possible state, when none of the other states are set. Further details on health checks are given in section 12.2.4.

**no data And Measurables**

If no measurements are carried out, but a sample value needs to be saved, then the sample value is set to no data for a measurable. This is a defined value, not a null data value. metrics and enummetrics can therefore also take the no data value.

**Entities And Measurables**

An entity is a concept introduced in Bright Cluster Manager version 8.0. Normally, a device, or a category or some similar grouping is a convenient idea to keep in mind as an entity, for concreteness.

The default entities in a new installation of Bright Cluster Manager are the following:

<table>
<thead>
<tr>
<th>entity</th>
</tr>
</thead>
<tbody>
<tr>
<td>device category partition</td>
</tr>
<tr>
<td>softwareimages</td>
</tr>
<tr>
<td>category ceph cloud csub</td>
</tr>
<tr>
<td>configuration overlay device</td>
</tr>
<tr>
<td>edge site etcd fspart group</td>
</tr>
<tr>
<td>job queue</td>
</tr>
<tr>
<td>jobs kubernetes network</td>
</tr>
<tr>
<td>node group openstack</td>
</tr>
<tr>
<td>partition profile rack</td>
</tr>
<tr>
<td>softwareimage user</td>
</tr>
</tbody>
</table>

For example, a software image object that is to be provisioned to a node is an entity, with some of the possible attributes of the entity being the name, kernelversion, creationtime, or locked attributes of the image:

```
[root@bright82 ~]# cmsh -c "softwareimage use default-image; show"
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>---------</td>
<td>-------</td>
</tr>
</tbody>
</table>
Because measurements can be carried out on such a variety of entities, it means that the monitoring 
and conditional actions that can be carried out on a Bright Cluster Manager cluster can be very diverse.
This makes entities a powerful and versatile concept in Bright Cluster Manager's monitoring system for 
managing clusters.

**Listing Measurables Used By An Entity**

In cmsh, for an entity, such as a device within device mode, a list of the measurables used by that device 
can be viewed with the measurables command.

**Example**

```
[bright82->device]% measurables node001
```

<table>
<thead>
<tr>
<th>Type</th>
<th>Name</th>
<th>Parameter</th>
<th>Class</th>
<th>Producer</th>
</tr>
</thead>
<tbody>
<tr>
<td>Enum</td>
<td>DeviceStatus</td>
<td></td>
<td>Internal</td>
<td>DeviceState</td>
</tr>
<tr>
<td>HealthCheck</td>
<td>ManagedServicesOk</td>
<td></td>
<td>Internal</td>
<td>CMDaemonState</td>
</tr>
<tr>
<td>HealthCheck</td>
<td>defaultgateway</td>
<td></td>
<td>Network</td>
<td>defaultgateway</td>
</tr>
<tr>
<td>HealthCheck</td>
<td>diskspace</td>
<td></td>
<td>Disk</td>
<td>diskspace</td>
</tr>
<tr>
<td>HealthCheck</td>
<td>dmesg</td>
<td></td>
<td>OS</td>
<td>dmesg</td>
</tr>
</tbody>
</table>

The subsets of these measurables—enummetrics, metrics, and health checks—can be listed with the 
enummetrics (section 12.2.2), metrics (section 12.2.3), or healthchecks (section 12.2.4) command.

In Bright View, all the entities that are using health checks can be viewed via the clickpath:

```
Monitoring→All Health Checks
```

**Listing Measurables From monitoring Mode**

Similarly, under monitoring mode, within the measurable submode, the list of measurable objects that 
can be used can be viewed with a list command:

**Example**

```
[bright82->monitoring]% measurable list
```

<table>
<thead>
<tr>
<th>Type</th>
<th>Name (key)</th>
<th>Parameter</th>
<th>Class</th>
<th>Producer</th>
</tr>
</thead>
<tbody>
<tr>
<td>Enum</td>
<td>DeviceStatus</td>
<td></td>
<td>Internal</td>
<td>DeviceState</td>
</tr>
<tr>
<td>HealthCheck</td>
<td>ManagedServicesOk</td>
<td></td>
<td>Internal</td>
<td>CMDaemonState</td>
</tr>
<tr>
<td>HealthCheck</td>
<td>Mon::Storage</td>
<td></td>
<td>Internal/Monitoring/Storage</td>
<td>MonitoringSystem</td>
</tr>
<tr>
<td>HealthCheck</td>
<td>chrootprocess</td>
<td></td>
<td>OS</td>
<td>chrootprocess</td>
</tr>
<tr>
<td>HealthCheck</td>
<td>cmsh</td>
<td></td>
<td>Internal</td>
<td>cmsh</td>
</tr>
</tbody>
</table>

© Bright Computing, Inc.
The subsets of these measurables—enummetrics, metrics, and health checks—can be listed with: list enum (section 12.2.2), list metric (section 12.2.3), or list healthcheck (section 12.2.4).
In Bright View, the equivalent to listing the measurables can be carried out via the clickpath:

Monitoring configuration→Measurables (figure 12.10, section 12.4.2)

and listing the subsets of the measurables is possible using column filtering (figure 12.11, section 12.4.2).

Viewing Parameters For A Particular Measurable From monitoring Mode
Within the measurable submode, parameters for a particular measurable can be viewed with the show command for that particular measurable:

Example

```
[bright82->monitoring->measurable]% use devicestatus
[bright82->monitoring->measurable(DeviceStatus)]% show
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Class</td>
<td>Internal</td>
</tr>
<tr>
<td>Consolidator</td>
<td>none</td>
</tr>
<tr>
<td>Description</td>
<td>The device status</td>
</tr>
<tr>
<td>Disabled</td>
<td>no (DeviceState)</td>
</tr>
<tr>
<td>Maximal age</td>
<td>0s (DeviceState)</td>
</tr>
<tr>
<td>Maximal samples</td>
<td>4,096 (DeviceState)</td>
</tr>
<tr>
<td>Name</td>
<td>DeviceStatus</td>
</tr>
<tr>
<td>Parameter</td>
<td></td>
</tr>
<tr>
<td>Producer</td>
<td>DeviceState</td>
</tr>
<tr>
<td>Revision</td>
<td></td>
</tr>
<tr>
<td>Type</td>
<td>Enum</td>
</tr>
</tbody>
</table>

12.2.2 Enummetrics
An enummetric is a measurable for an entity that can only take a limited set of values. At the time of writing (June 2017), DeviceStatus is the only enummetric. This may change in future versions of Bright Cluster Manager.

The full list of possible values for the enummetric DeviceStatus is:
up, down, closed, installing, installer_failed, installer_rebooting, installer_callinginit, installer_unreachable, installer_burning, burning, unknown, opening, going_down, pending, and no data.

The enummetrics available for use can be listed from within the measurable submode of the monitoring mode:

Example

```
[bright82->monitoring->measurable]% list enum
```

<table>
<thead>
<tr>
<th>Type</th>
<th>Name (key)</th>
<th>Parameter</th>
<th>Class</th>
<th>Producer</th>
</tr>
</thead>
<tbody>
<tr>
<td>Enum</td>
<td>DeviceStatus</td>
<td>Internal</td>
<td>DeviceState</td>
<td></td>
</tr>
</tbody>
</table>

The list of enummetrics that is configured to be used by an entity, such as a device, can be viewed with the enummetrics command for that entity:

Example
The states that the entity has been through can be viewed with a dumpmonitoringdata command (section 12.6.4):

Example

```
[bright82->device]% dumpmonitoringdata -99d now devicestatus node001
```

Timestamp Value Info
-------------------------- ----------- ----------
2017/07/03 16:07:00.001 down
2017/07/03 16:09:00.001 installing
2017/07/03 16:09:29.655 no data
2017/07/03 16:11:00 up
2017/07/12 16:05:00 up

The parameters of an enummetric such as devicestatus can be viewed and set from monitoring mode, from within the measurable submode (page 464).

12.2.3 Metrics

A metric for an entity is typically a numeric value for an entity. The value can have units associated with it.

In the basic example of section 12.1, the metric value considered was CPUUser, measured at the default regular time intervals of 120s.

The value can also be defined as no data. no data is substituted for a null value when there is no response for a sample. no data is not a null value once it has been set. This means that there are no null values stored for monitored data.

Other examples for metrics are:

- LoadOne (value is a number, for example: 1.23)
- WriteTime (value in ms/s, for example: 5 ms/s)
- MemoryFree (value in readable units, for example: 930 MiB, or 10.0 GiB)

A metric can be a built-in, which means it comes with Bright Cluster Manager as integrated code within CMDaemon. This is based on c++ and is therefore much faster than the alternative. The alternative is that a metric can be a standalone script, which means that it typically can be modified more easily by an administrator with scripting skills.

The word metric is often used to mean the script or object associated with a metric as well as a metric value. The context makes it clear which is meant.

A list of metrics in use can be viewed in cmsh using the list command from monitoring mode:

Example

```
[bright82->monitoring]% measurable list metric
```

Type Name (key) Parameter Class Producer
------- ------------------------ -------------- ------------------ -------------
Metric AlertLevel count Internal AlertLevel
Metric AlertLevel maximum Internal AlertLevel
...
In Bright View, the metrics can be viewed with the clickpath:

Monitoring configuration → Measurables

and then clicking on the filter widget to select Metric (figure 12.10, section 12.4.2).

A list of metrics in use by an entity can be viewed in cmsh using the metrics command for that entity. For example, for the entity node001 in mode devices:

Example

[bright82->devices]% metrics node001

<table>
<thead>
<tr>
<th>Type</th>
<th>Name</th>
<th>Parameter</th>
<th>Class</th>
<th>Producer</th>
</tr>
</thead>
<tbody>
<tr>
<td>Metric</td>
<td>AlertLevel</td>
<td>count</td>
<td>Internal</td>
<td>AlertLevel</td>
</tr>
<tr>
<td>Metric</td>
<td>AlertLevel</td>
<td>maximum</td>
<td>Internal</td>
<td>AlertLevel</td>
</tr>
</tbody>
</table>

... The parameters of a metric such as AlertLevel:count can be viewed and set from monitoring mode, from within the measurable submode, just as for the other measurables:

Example

[bright82->monitoring->measurable]% use alertlevel:count
[bright82->monitoring->measurable[AlertLevel:count]]% show

Parameter                  Value
-----------------------------------------------
Class           Internal
Consolidator    default
Cumulative      no
Description     Number of active triggers
Disabled        no
Maximal age     0s
Maximal samples 0
Maximum         0
Minimum         0
Name            AlertLevel
Parameter       count
Producer        AlertLevel
Revision        
Type            Metric

The equivalent Bright View clickpath to edit the parameters is:

Monitoring configuration → Measurables → Edit

12.2.4 Health Check

A health check value is a response to a check carried out on an entity. The response indicates the health of the entity for the check that is being carried out.

For example, the ssh2node health check, which runs on the head node to check if the SSH port 22 access to regular nodes is reachable.

A health check is run at a regular time interval, and can have the following possible values:

- **PASS**: The health check succeeded. For example, if ssh2node is successful, which suggests that an ssh connection to the node is fine.
12.2 Monitoring Concepts And Definitions

- **FAIL**: The health check failed. For example, if ssh2node was rejected. This suggests that the ssh connection to the node is failing.

- **UNKNOWN**: The health check did not succeed, did not fail, but had an unknown response. For example, if ssh2node has a timeout, for example due to routing or other issues. It means that it is unknown whether the connection is fine or failing, because the response that came in is unknown. Typically the administrator should investigate this further.

- **no data**: The health check did not run, so no data was obtained. For example, if ssh2node is disabled for some time, then no data values were obtained during this time. Since the health check is disabled, it means that no data cannot be recorded during this time by ssh2node. However, because having a no data value in the monitoring data for this situation is a good idea—explicitly knowing about having no data is helpful for various reasons—then no data values can be set, by CMDaemon, for samples that have no data.

Other examples of health checks are:

- **diskspace**: check if the hard drive still has enough space left on it
- **mounts**: check mounts are accessible
- **mysql**: check status and configuration of MySQL is correct
- **hpraid**: check RAID and health status for certain HP RAID hardware

These and others can be seen in the directory: `/cm/local/apps/cmd/scripts/healthchecks`.

**Health Checks**

In Bright View, the health checks that can be configured for all entities can be seen with the clickpath:

```
Monitoring configuration → Measurables
```

(figure 12.10, section 12.4.2)

and then clicking on the filter widget to select **Health Check** (figure 12.11, section 12.4.2). Options can be set for each health check by clicking through via the **Edit** button.

**All Configured Health Checks**

In Bright View, health checks that have been configured for all entities can be seen with the clickpath:

```
Monitoring configuration → All Health Checks
```

(section 12.4.7)

The view can be filtered per column.

**Configured Health Checks For An Entity**

An overview can be seen for a particular entity `<entity>` via the clickpath:

```
Monitoring configuration → Health status → `<entity>` → Show
```

**Severity Levels For Health Checks, And Overriding Them**

A health check has a settable severity (section 12.2.7) associated with its response defined in the trigger options.

For standalone healthchecks, the severity level defined by the script overrides the value in the trigger. For example, FAIL 40 or UNKNOWN 10, as is set in the hpraid health check (`/cm/local/apps/cmd/scripts/healthchecks/hpraid`).

Severity values are processed for the **AlertLevel** metric (section 12.2.8) when the health check runs.

© Bright Computing, Inc.
Default Templates For Health Checks And Triggers

A health check can also launch an action based on any of the response values.

Monitoring triggers have the following default templates:

- **Failing health checks**: With a default severity of 15
- **Passing health checks**: With a default severity of 0
- **Unknown health checks**: With a default severity of 10

The severity level is one of the default parameters for the corresponding health checks. These defaults can also be modified to allow an action to be launched when the trigger runs, for example, sending an e-mail notification whenever any health check fails.

With the default templates, the actions are by default set for all health checks. However, specific actions that are launched for a particular measurable instead of for all health checks can be configured. To do this, one of the templates can be cloned, the trigger can be renamed, and an action can be set to launch from a trigger. The reader should be able to recognize that in the basic example of section 12.1 this is how, when the metric measurable CPUUser crosses 50 jiffies, the killallyesttrigger is activated, and the killallyyes action script is run.

### 12.2.5 Trigger

A **trigger** is a threshold condition set for a sampled measurable. When a sample crosses the threshold condition, it enters or leaves a zone that is demarcated by the threshold.

A trigger zone also has a settable severity (section 12.2.7) associated with it. This value is processed for the *AlertLevel* metric (section 12.2.8) when an action is triggered by a threshold event.

Triggers are discussed further in section 12.4.5.

### 12.2.6 Action

In the basic example of section 12.1, the action script is the script added to the monitoring system to kill all *yes* processes. The script runs when the condition is met that CPUUser crosses 50 jiffies.

An **action** is a standalone script or a built-in command that is executed when a condition is met, and has exit code 0 on success. The condition that is met can be:

- A FAIL, PASS, UNKNOWN, or no data from a health check
- A trigger condition. This can be a FAIL or PASS for conditional expressions.
- State flapping (section 12.2.9).

The actions that can be run are listed from within the action submode of the monitoring mode.

#### Example

```
[bright82->monitoring->action]% list
Type Name (key) Run on Action
----------- ---------------- ----------------------------------------------------------
Drain Drain Active Drain node from all WLM
Email Send e-mail Active Send e-mail
Event Event Active Send an event to users with connected client
ImageUpdate ImageUpdate Active Update the image on the node
PowerOff PowerOff Active Power off a device
PowerOn PowerOn Active Power on a device
PowerReset PowerReset Active Power reset a device
Reboot Reboot Node Reboot a node
Script killallyesaction Node /cm/local/apps/cmd/scripts/actions/killallyyes
Script killprocess Node /cm/local/apps/cmd/scripts/actions/killprocess.pl
```

© Bright Computing, Inc.
12.2 Monitoring Concepts And Definitions

Script remount Node /cm/local/apps/cmd/scripts/actions/remount
Script testaction Node /cm/local/apps/cmd/scripts/actions/testaction
Shutdown Shutdown Node Shutdown a node
Undrain Undrain Active Undrain node from all WLM

The Bright View equivalent is accessible via the clickpath:

Monitoring configuration→Actions (figure 12.17, section 12.4.4)

Configuration of monitoring actions is discussed further in section 12.4.4.

12.2.7 Severity

Severity is a positive integer value that the administrator assigns for a trigger. It takes one of these 6 suggested values:

<table>
<thead>
<tr>
<th>Value</th>
<th>Name</th>
<th>Icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>debug</td>
<td>📖</td>
<td>debug message</td>
</tr>
<tr>
<td>0</td>
<td>info</td>
<td>📖</td>
<td>informational message</td>
</tr>
<tr>
<td>10</td>
<td>notice</td>
<td>📖</td>
<td>normal, but significant, condition</td>
</tr>
<tr>
<td>20</td>
<td>warning</td>
<td>📖</td>
<td>warning conditions</td>
</tr>
<tr>
<td>30</td>
<td>error</td>
<td>📖</td>
<td>error conditions</td>
</tr>
<tr>
<td>40</td>
<td>alert</td>
<td>📖</td>
<td>action must be taken immediately</td>
</tr>
</tbody>
</table>

Severity levels are used in the AlertLevel metric (section 12.2.8). They can also be set by the administrator in the return values of health check scripts (section 12.2.4).

By default the severity value is 15 for a health check FAIL response, 10 for a health check UNKNOWN response, and 0 for a health check PASS response (section 12.2.4).

12.2.8 AlertLevel

AlertLevel is a special metric. It is sampled and re-calculated when an event with an associated Severity (section 12.2.7) occurs. There are three types of AlertLevel metrics:

1. AlertLevel (count): the number of events that are at notice level and higher. The aim of this metric is to alert the administrator to the number of issues.
2. AlertLevel (max): simply the maximum severity of the latest value of all the events. The aim of this metric is to alert the administrator to the severity of the most important issue.
3. AlertLevel (sum): the sum of the latest severity values of all the events. The aim of this metric is to alert the administrator to the overall severity of issues.

12.2.9 Flapping

Flapping, or State Flapping, is when a measurable trigger that is detecting changes, (section 12.4.5) changes that are too frequent. That is, the measurable goes in and out of the zone too many times over a number of samples. In the basic example of section 12.1, if the CPUUser metric crossed the threshold zone 5 times within 5 minutes (the default values for flap detection), then it would by default be detected as flapping. A flapping alert would then be recorded in the event viewer, and a flapping action could also be launched if configured to do so.

12.2.10 Data Producer

A data producer produces measurables. Sometimes it can be a group of measurables, as in the measurables provided by a data producer that is being used:

© Bright Computing, Inc.
### Example

```
% list -f name:25,producer:15 | grep ProcStat
```

<table>
<thead>
<tr>
<th>Name</th>
<th>Measurables</th>
</tr>
</thead>
<tbody>
<tr>
<td>BlockedProcesses</td>
<td>ProcStat</td>
</tr>
<tr>
<td>CPUGuest</td>
<td>ProcStat</td>
</tr>
<tr>
<td>CPUIdle</td>
<td>ProcStat</td>
</tr>
<tr>
<td>CPUIrq</td>
<td>ProcStat</td>
</tr>
<tr>
<td>CPUNice</td>
<td>ProcStat</td>
</tr>
<tr>
<td>CPUSoftIrq</td>
<td>ProcStat</td>
</tr>
<tr>
<td>CPUSTeal</td>
<td>ProcStat</td>
</tr>
<tr>
<td>CPUSystem</td>
<td>ProcStat</td>
</tr>
<tr>
<td>CPUUser</td>
<td>ProcStat</td>
</tr>
<tr>
<td>CPUWait</td>
<td>ProcStat</td>
</tr>
<tr>
<td>CtxtSwitches</td>
<td>ProcStat</td>
</tr>
<tr>
<td>Forks</td>
<td>ProcStat</td>
</tr>
<tr>
<td>Interrupts</td>
<td>ProcStat</td>
</tr>
<tr>
<td>RunningProcesses</td>
<td>ProcStat</td>
</tr>
</tbody>
</table>

Sometimes it may just be one measurable, as provided by a used data producer:

### Example

```
% list -f name:25,producer:15 | grep ssh2node
```

```
ssh2node    ssh2node
```

It can even have no measurables, and just be an empty container for measurables that are not in use yet.

In cmsh all possible data producers (used and unused) can be listed as follows:

### Example

```
% list
```

The equivalent in Bright View is via the clickpath:

Monitoring Configuration → Data Producers

The data producers configured for an entity, such as a head node bright82, can be listed with the `monitoringproducers` command:

### Example

```
% monitoringproducers
```

<table>
<thead>
<tr>
<th>Type</th>
<th>Name</th>
<th>Arguments</th>
<th>Measurables</th>
<th>Node execution filters</th>
</tr>
</thead>
<tbody>
<tr>
<td>AlertLevel</td>
<td>AlertLevel</td>
<td>3 / 231</td>
<td>&lt;0 in submode&gt;</td>
<td></td>
</tr>
<tr>
<td>CMDaemonState</td>
<td>CMDaemonState</td>
<td>1 / 231</td>
<td>&lt;0 in submode&gt;</td>
<td></td>
</tr>
<tr>
<td>ClusterTotal</td>
<td>ClusterTotal</td>
<td>18 / 231</td>
<td>&lt;1 in submode&gt;</td>
<td></td>
</tr>
<tr>
<td>Collection</td>
<td>NFS</td>
<td>32 / 231</td>
<td>&lt;0 in submode&gt;</td>
<td></td>
</tr>
<tr>
<td>Collection</td>
<td>sdt</td>
<td>0 / 231</td>
<td>&lt;0 in submode&gt;</td>
<td></td>
</tr>
<tr>
<td>DeviceState</td>
<td>DeviceState</td>
<td>1 / 231</td>
<td>&lt;1 in submode&gt;</td>
<td></td>
</tr>
<tr>
<td>HealthCheckScript</td>
<td>chrootprocess</td>
<td>1 / 231</td>
<td>&lt;1 in submode&gt;</td>
<td></td>
</tr>
<tr>
<td>HealthCheckScript</td>
<td>cmsh</td>
<td>1 / 231</td>
<td>&lt;1 in submode&gt;</td>
<td></td>
</tr>
<tr>
<td>HealthCheckScript</td>
<td>defaultgateway</td>
<td>1 / 231</td>
<td>&lt;0 in submode&gt;</td>
<td></td>
</tr>
<tr>
<td>HealthCheckScript</td>
<td>diskspace</td>
<td>1 / 231</td>
<td>&lt;0 in submode&gt;</td>
<td></td>
</tr>
<tr>
<td>HealthCheckScript</td>
<td>dmesg</td>
<td>1 / 231</td>
<td>&lt;0 in submode&gt;</td>
<td></td>
</tr>
</tbody>
</table>

© Bright Computing, Inc.
The displayed data producers are the ones configured for the entity, even if there are no measurables used by the entity.

Data producer configuration in Bright View is discussed further in section 12.4.1.

Access Control For Monitoring Data

Access control to data producers: An access control setting for a data producer determines who can plot (via the measurable monitoring interface used by Bright View and the User Portal), or view data (using the text-based interface of cmsh or pythoncm) from the measurables generated by a data producer. Thus, for example, the charts in the user portal (section 12.7) can be restricted according to the data producer that generates them.

There are three possible settings for access control for data producers. If the data producer is set to:

1. Public: then it means any user can, by default, plot/view data derived from that data producer. This is because by default a user has the token PLOT_TOKEN in their profile.

2. Private: then it means that a non-root user cannot, by default, plot/view data derived from that data producer. This is because, by default, non-root users do not have the token PRIVATE_MONITORING_TOKEN in their profile. If that token is in the profile, then the user has an elevated privilege, and can plot/view data, just like root.

3. Individual: then it means that a non-root user, by default, can plot/view the data only if the job associated with that data was run by that same non-root user. More verbosely, with the default user settings: the user who ran a job for which job measurables are produced by a data producer, must be the same as the user that who wants to plot/view the data, or else the data
cannot be plotted/viewed. The exception to this is, as already suggested, if the user that wants to plot/view the data is root, or has a user profile with the token PRIVATE_MONITORING_TOKEN. In that case the data can be plotted/viewed.

On a regular Bright Cluster Manager cluster, only a few low level data producers are set to private. An administrator can decide to set a data producer access control value to one of the three possible values, by using the setup submode of monitoring mode:

**Example**

```
[root@bright82 ~]# cmsh
[bright82]% monitoring setup
[bright82->monitoring->setup]% use mounts
[bright82->...[mounts]]% set access private
[bright82->...*[mounts*]]% commit
```

If a data producer is newly added, then by default its access control value is set to public. Changing this to private at a later time means that access to past and future data values from that data producer are affected by the private setting. If access is changed once more back to public, then it means that access to past and future data values are once again viewable and plottable by all users.

The current settings for access control for the data producers can be seen with:

**Example**

```
[root@bright82 ~]# cmsh -c "monitoring setup; list -f name:40,access"
```

<table>
<thead>
<tr>
<th>name (key)</th>
<th>access</th>
</tr>
</thead>
<tbody>
<tr>
<td>AggregateNode</td>
<td>Public</td>
</tr>
<tr>
<td>AlertLevel</td>
<td>Public</td>
</tr>
<tr>
<td>BigDataTools</td>
<td>Public</td>
</tr>
<tr>
<td>CMDaemonState</td>
<td>Public</td>
</tr>
<tr>
<td>Cassandra</td>
<td>Public</td>
</tr>
<tr>
<td>ClusterTotal</td>
<td>Public</td>
</tr>
<tr>
<td>...</td>
<td></td>
</tr>
</tbody>
</table>

Access control to measurables: Measurables can also have access controls.

Access control for measurables is by default inherited from the data producer that generates it. It can be overwritten:

**Example**

```
[root@bright82 ~]# cmsh
[bright82]% monitoring measurable
[bright82->monitoring->measurable]% use loadone
[bright82->...[LoadOne]]% get access
Public (SysInfo)
[bright82->...*[LoadOne*]]% set access private
[bright82->...*[LoadOne*]]% commit
```

---

1 Even more verbosely: Individual access control is meant for job-based measurables, and works like this:
  
  All monitoring data is stored per (entity, measurable) pair.
  
  If the measurable has an access control value of individual, then a user check is performed. If the login name is the same as the user that owns the entity, then data can be plotted/viewed. If the user check does not match, then no data is returned.
  
  Jobs—which are entities—are owned by the user that ran the job. Similarly Prometheus (entities) can have a 'user="Alice"' label set, to define ownership. No other entity managed by Bright Cluster Manager is owned by a user.
  
  For all unowned entities, individual access is equivalent to private access.
A measurable can thus take an access control value of Public, Private, or Individual. It can also explicitly be set to a value of inherit, which sets it to the value of its data producer. The inheritance is indicated in cmsh by enclosing the parent data producer in parentheses, as shown in the preceding example.

The current settings for access control for the measurables can be seen with:

Example

```
[root@bright82 ~]# cmsh -c "monitoring measurable; list -f name:40,access"
name (key) access
---------------------------------------- --------------------
AlertLevel Public
AlertLevel Public
AlertLevel Public
AvgJobDuration Public
BlockedProcesses Public
... 
```

12.2.11 Conceptual Overview: The Main Monitoring Interfaces Of Bright View

Bright View has its own display modes, event and action results logs, and account handling screen. These can be selected via some of the 8 icons in the top right corner of the Bright View standard display (figure 12.5):

The 8 icons are described from left to right next:

1. **Settings** mode is the display that is shown when Bright View first starts up.

   Within **Settings** mode is the **Monitoring Configuration** resource. This resource should not be confused with the Bright View **Monitoring** mode, which is launched by the next icon in figure 12.5. **Monitoring configuration** is about configuring how items are monitored and how their data values are collected, and is discussed further in section 12.4.

2. The **Monitoring** mode allows visualization of the data values collected according to the specifications of the Bright View **Settings** mode. The visualization allows graphs to be configured, and is discussed further in section 12.3.

3. The **Accounting** mode typically allows visualization of job resources used by users, although it can be used to visualize job resources used by other aggregation entities. This is helpful tracking resources consumed by users. **Job accounting** is discussed further in section 12.9.

4. The **Events** icon allows logs of events (section 12.10.1) to be viewed.

5. The **Action results** icon allows the logs of the results of actions to be viewed.

6. The **Background tasks** icon allows background tasks to be viewed.
7. The Unsaved entities icon allows entities that have not yet been saved to be viewed.

8. The Account handling icon allows account settings to be managed for the Bright View user.

The monitoring aspects of the first two icons are discussed in greater detail in the sections indicated.

12.3 Monitoring Visualization With Bright View

The Monitoring icon in the menu bar of Bright View (item 2 in figure 12.5) launches an intuitive visualization tool that is the main GUI tool for getting a feel of the system’s behavior over periods of time. With this tool the measurements and states of the system can be viewed as resizable and overlayable graphs. The graphs can be zoomed in and out on over a particular time period, the graphs can be laid out on top of each other or the graphs can be laid out as a giant grid. The graph scale settings can also be adjusted, stored and recalled for use the next time a session is started.

An alternative to Bright View’s visualization tool is the command-line cmsh. This has the same functionality in the sense that data values can be selected and studied according to configurable parameters with it (section 12.6). The data values can even be plotted and displayed on graphs with cmsh with the help of unix pipes and graphing utilities. However, the strengths of monitoring with cmsh lie elsewhere: cmsh is more useful for scripting or for examining pre-decided metrics and health checks rather than a quick visual check over the system. This is because cmsh needs more familiarity with options, and is designed for text output instead of interactive graphs. Monitoring with cmsh is discussed in sections 12.5 and 12.6.

Visualization of monitoring graphs with Bright View is now described.

12.3.1 The Monitoring Window

If the Monitoring icon is clicked on from the menu bar of Bright View (figure 12.5), then a monitoring window for visualizing data opens up. By default, this displays blank plot panels—graph axes with a time scale going back some time on the $x$-axis, and with no $y$-axis measurable data values plotted (figure 12.6).

![Bright View Monitoring Window: Blank Plot Panel](image)

Finding And Selecting The Measurable To Be Plotted

To plot measurables, the entity which it belongs to should be selected from the navigation menu on the left-hand side. Once that has been selected, a class for that measurable can be chosen, and then
the measurable itself can be selected. For example, to plot the measurable CPUUser for a head node bright82, it can be selected from the navigation clickpath Device→bright82→CPU→CPUUser.

Sometimes, finding a measurable is easier with the Search box. Typing in CPUUser there shows all the measurables with that text (figure 12.7). The search is case-insensitive.

![Figure 12.7: Bright View Monitoring Window: Search Box In Navigation](image)

The search box can handle some simple regexes too, with .* and | taking their usual meaning:

**Example**

- node001.*cpuuser: select a measurable with a data path that starts with node001 and ends with cpuuser, with 0 or more characters of any kind in between.
- (node001|node002).*cpuuser: as for preceding example, but including node002 as an alternative to node001.

The / (forward slash) allows filtering according to the data path. It corresponds to the navigation depth in the tree hierarchy:

**Example**

- node001/cpu/cpuuser: search for a measurable with a data path that matches node001/cpu/cpuuser

**Plotting The Measurable**

Once the measurable is selected, it can be drag-and-dropped into a plot panel. This causes the data values to be plotted.

When a measurable is plotted into a panel, two graph plots are displayed. The smaller, bottom plot, represents the polled value as a bar chart. The larger, upper plot, represents an interpolated line graph. Different kinds of interpolations can be set. To get a quick idea of the effect of different kinds of interpolations, https://bl.ocks.org/mbostock/4342190 is an interactive overview that shows how they work on a small set of values.

The time axes can be expanded or shrunk using the mouse wheel in the graphing area of the plot panel. The resizing is carried out centered around the position of the mouse pointer.

© Bright Computing, Inc.
12.4 Monitoring Configuration With Bright View

This section is about the configuration of monitoring for measurables, and about setting up trigger actions.

If Bright View is running in the standard Settings mode, which is the first icon in figure 12.5, page 473, then selecting Monitoring Configuration from the resources section makes the following menu items available for managing or viewing:

- Data Producers (section 12.4.1)
- Measurables (section 12.4.2)
- Consolidators (section 12.4.3)
- Actions (section 12.4.4)
- Triggers (section 12.4.5)
- Health status (section 12.4.6)
- All Health checks (section 12.4.7)
- Standalone Monitored Entities (section 12.4.8)

![Figure 12.8: Bright View Monitoring Configuration Settings](image)

These settings (figure 12.8) are now discussed in detail.

12.4.1 Monitoring Configuration: Data Producers

The Data Producers window lists all the data producers. Data producers are introduced in section 12.2.10.

Each data producer can have its settings edited within a subwindow. For example, the ProcStat data producer, which produces data for several measurables, including CPUUser, has the settings shown in figure 12.9:

© Bright Computing, Inc.
When the data producer takes samples to produce data, run length encoding (RLE) is used to compress the number of samples that are stored as data. Consolidation is carried out on the RLE samples. Consolidation in Bright Cluster Manager means gathering several data values, and making one value from them over time periods. Consolidation is done as data values are gathered. The point at which data values are discarded, if ever, is thus not dependent on consolidation.

Some remarks about some of the data producer settings that are seen in the subwindow of figure 12.9:

- **Maximal samples**: the maximum number of RLE samples that are kept. If set to 0, then the number of samples is not considered.

- **Maximal Age**: the maximum age of RLE samples that are kept. If Maximal Age is set to 0 then the sample age is not considered.

With Maximal samples and Maximal Age, the first of the rules that is reached is the one that causes the exceeding RLE samples to be dropped.

Samples are kept forever if Maximal samples and Maximal Age are both set to 0. This is discouraged due to the risk of exceeding the available data storage space.

- **Interval**: the interval between sampling, in seconds.

- **Offset**: A time offset from start of sampling. Some sampling depends on other sampling to be carried out first. This is used, for example, by data producers that rely on sampling from other data producers. For example, the AggregateNode data producer, which has measurables such as TotalCPUIdle and TotalMemoryFree. The samples for AggregateNode depend upon the ProcStat data producer, which produces the CPUIdle measurable; and the ProcMemInfo data producer, which produces the MemoryFree measurable.

- **Fuzzy offset**: a multiplier in the range from 0 to 1. It is multiplied against the sampling time interval to fix a maximum value for the time offset for when the sampling takes place. The actual offset used per node is spread out reasonably evenly within the range up to that maximum time offset.
For example, for a sampling time interval of 120s:

If the offset is 0, then there is no offset, and the sampling is attempted for all nodes at time instant when the interval restarts. This can lead to an overload at the time of sampling.

If, on the other hand, the offset is 0.25, then the sampling is done within a range offset from the time of sampling by a maximum of $0.25 \times 120\text{s} = 30\text{s}$. So, each node is sampled at a time that is offset by up to 30s from when the 120s interval restarts. From the time the change in value of the fuzzy offset starts working, the offset is set for each node. The instant at which sampling is carried out on a node then differs from the other nodes, even though each node still has an interval of 120s between sampling. An algorithm is used that tends to even out the spread of the instants at which sampling is carried out within the 30s range. The spreading of sampling has the effect of reducing the chance of overload at the time of sampling.

- **Consolidator:** By default, set to the default group. The default group consolidates (summarizes) the RLE samples over periods of an hour, a day, and a week. Consolidators are explained further in section 12.4.3.

- **Node execution filters:** A way to filter execution (restrict execution) of the data producer. It tells Bright Cluster Manager where the data producer runs. If not set, then the data producer runs on all nodes managed by CMDaemon. Filters can be for nodes, types, overlays, resources, and categories.

- **Execution multiplexer:** A way to multiplex execution (have execution work elsewhere) for a data producer. It tells Bright Cluster Manager about the entities that the data producer is sampling for. A data producer gathers data at the nodes defined by the node execution filter, and with multiplex execution the data producer gathers samples from other entities. These entities can be nodes, types, overlays, and resources. The entities from which it can sample are defined into groups called execution multiplexers. Execution multiplexers can thus be node multiplexers, type multiplexers, overlay multiplexers, or resource multiplexers.

- **When:** This has three possible values:
  - **Timed:** Data producer is run at a periodic Interval. This is the default.
  - **Pre job:** Data producer is only run before a job, in the prolog of the job. This is typically a check as part of the workload manager process (section 7.8.2).
  - **On demand:** Data producer is only run on demand, and not at a periodic Interval.

- **Only when idle:** By default a data producer runs regardless of how busy the nodes are. However, if the Only when idle setting is enabled, then the data producer runs only when the node is idle. Idle is a condition that is defined by the metric condition $\text{LoadOne}>1$ (page 700).

### 12.4.2 Monitoring Configuration: Measurables

The Measurables window lists the available measurables (figure 12.10):
There are many measurables. It can therefore be useful to use the filtering-by-column option at the top of the columns. Each filtering option also has a search box associated with it (figure 12.11):

From the measurables window, a subwindow can be opened with the Edit button for a measurable. This accesses the options for a particular measurable (figure 12.12):
The options shown include the sampling options: Maximal age, Maximal samples, Consolidator. The sampling options work as described for data producers (section 12.4.1).

Other options for a metric are setting the Maximum and Minimum values, the Unit used, and whether the metric is Cumulative.

If a metric is cumulative, then it is monotonic. Monotonic means that the metric only increments (is cumulative), as time passes. In other words, if the metric is plotted as a graph against time, with time on the x-axis, then the metric never descends. Normally the increments are from the time of boot onwards, and the metric resets at boot. For example, the number of bytes received at an interface is cumulative, and resets at boot time.

Usually the cluster administrator is only interested in the differential value of the metric per sample interval. That is, the change in the value of the current sample, from its value in the preceding sample. For example, bytes/second, rather than total number of bytes up to that time from boot.

**12.4.3 Monitoring Configuration: Consolidators**

**Introduction To Consolidators**

The concept of consolidators is explained using simple ascii graphics in Appendix K, while the cmsh interface to the consolidators submode is discussed in section 12.5.2.

In this current section, the Bright View interface to consolidators is discussed.

In Bright View, the Consolidators window lists all consolidator groups (figure 12.13).
Subwindows allow the component consolidators to be created or modified (figure 12.14).

There are two pre-existing consolidator groups: **default** and **none**.

**The none Consolidator Group**
The none consolidator group, has no consolidators. Using a consolidator of none for a measurable or data producer means that samples are not consolidated. This can be dangerous if the cluster is more likely to run out of space due to unrestrained sampling, as can occur, for example, if Maximal Age and Maximal samples (section 12.4.1) for data producers are both set to 0.

**The default Consolidator Group**
The default consolidator group consists of the consolidators hour, day, and week. These are, unsurprisingly, defined to consolidate the samples in intervals of an hour, day, or week.

A consolidated value is generated on-the-fly. So, for example, during the hour that samples of a measurable come in, the hour consolidator uses the samples to readjust the consolidated value for that hour. When the hour is over, the consolidated value is stored for that hour as the data value for that hour, and a new consolidation for the next hour begins.

Consolidator values are kept, as for sample values, until the Maximal Age and Maximal sample settings prevent data values being kept.

**Other Consolidator Group Possibilities**
Other sets of custom intervals can also be defined. For example, instead of the default consolidator group, (hour, day, week), a similar group called decimalminutes consolidator group, (1min, 10min, 100min, 1000min, 10000min) could be created with the appropriate intervals (figure 12.15):
Consolidator Item Settings

Consolidator items are members of the consolidator groups. The items have settings as properties, and can be managed (figure 12.16).

For the consolidator of hour, within the default consolidators group, the clickpath to edit its properties is:

```
Monitoring Configuration → Consolidators [default] → Edit → Consolidator [hour] → Edit
```

The properties that can be set for a consolidator item are:

- **Name**: The name of the consolidator item. By default, for the consolidator group default, the consolidator items with names of Day, Hour, and Month are already set up, with appropriate values for their corresponding fields.

- **Maximal samples**: The maximum number of samples that are stored for that consolidator item. This should not be confused with the maximal samples of the measurable being consolidated.

- **Interval**: The time period (in seconds) covered by the consolidator sample. For example, the consolidator with the name hour has a value of 3600. The property should not be confused with the time period between samples of the measurable being consolidated.

- **Offset**: The time offset from the default consolidation time, explained in more detail shortly.
• **Kind**: The kind of consolidation that is done on the raw data samples. The value of `kind` is set to average by default. The output result for a processed set of raw data—the consolidated data point—is an average, a maximum or a minimum of the input raw data values. `Kind` can thus have the value Average, Maximum, or Minimum. The value of `kind` is set to average by default.

For a given consolidator, when one `Kind` is changed to another, the historically processed data values become inconsistent with the newer data values being consolidated. Previous consolidated data values for that consolidator are therefore discarded during such a change.

To understand what `Offset` means, the **Maximal samples** of the measurable being consolidated can be considered. This is the maximum number of raw data points that the measurable stores. When this maximum is reached, the oldest data point is removed from the measurable data when a new data point is added. Each removed data point is gathered and used for data consolidation purposes.

For a measurable that adds a new data point every `Interval` seconds, the time $t_{raw\ gone}$, which is how many seconds into the past the raw data point is removed, is given by:

$$
t_{raw\ gone} = \text{Maximal}\ samples_{measurable} \times \text{Interval}_{measurable}
$$

This value is also the default consolidation time, because the consolidated data values are normally presented from $t_{raw\ gone}$ seconds ago, to further into the past. The default consolidation time occurs when the `Offset` has its default, zero value.

If however the `Offset` period is non-zero, then the consolidation time is offset, because the time into the past from which consolidation is presented to the user, $t_{consolidation}$, is then given by:

$$
t_{consolidation} = t_{raw\ gone} + \text{Offset}
$$

The monitoring visualization graphs then show consolidated data from $t_{consolidation}$ seconds into the past, to further into the past $^2$.

### 12.4.4 Monitoring Configuration: Actions

Actions are introduced in section 12.2.6. The **Actions** window (figure 12.17) displays actions that Bright Cluster Manager provides by default, and also displays any custom actions that have been created:

![Actions Window](image)

**Figure 12.17: Bright View Monitoring Configuration: Actions**

The `killallyes` script from the basic example of section 12.1 would show up here if it has been implemented.

Actions are triggered, by triggers (section 12.4.5).

By default, the following actions exist:

$^2$ For completeness: the time $t_{consolidation\ gone}$, which is how many seconds into the past the consolidated data goes and is viewable, is given by an analogous equation to that of the equation defining $t_{raw\ gone}$:

$$
t_{consolidation\ gone} = \text{Maximal}\ samples_{consolidation} \times \text{Interval}_{consolidation}
$$
• **Poweron**: Powers on the node
• **PowerOff**: Powers off the node
• **PowerReset**: Hard resets the node
• **Drain**: Drains the node (does not allow new jobs on that node)
• **Undrain**: Undrains the node (allows new jobs on that node)
• **Reboot**: Reboots node via the operating system
• **Shutdown**: Shuts the node down via the operating system
• **ImageUpdate**: Updates the node from the software image
• **Event**: Sends an event to users connected with cmsh or Bright View
• **killprocess**: A script to kill a process
• **remount**: A script to remount all devices
• **testaction**: A test script
• **Send e-mail to administrators**: Sends an e-mail out

The preceding actions show their options when the associated Edit button is clicked. A subwindow with options opens up. The following options are among those then displayed:

• **Run on**: What nodes the action should run on. Choices are:
  - active: the action runs on the active node only
  - node: the action runs on the triggering node

• **Allowed time**: The time interval in the 24 hour clock cycle that the action is allowed to start running. The interval can be restricted further to run within certain days of the week, months of the year, or dates of the month. Days and months must be specified in lower case.

Rather than defining a formal syntax, some examples are given, with explanations:

- **november-march**: November to March. June, July and August are forbidden.
- **november-march{monday-saturday}**: As in the preceding, but all Sundays are also forbidden.
- **november-march{monday-saturday{13:00-17:00}}**: Restricted to the period defined in the preceding example, and with the additional restriction that the action can start running only during the time 13:00-17:00.
- **09:00-17:00**: All year long, but during 09:00-17:00 only.
- **monday-friday{9:00-17:00}**: All year long, but during 9:00-17:00 only, and not on Saturdays or Sundays.
- **november-march{monday-saturday{13:00-17:00}}**: Not in June, July, and August. In the other months, only on Mondays to Saturdays, from 13:00-17:00.
- **may-september{monday-friday{09:00-18:00};saturday-sunday{13:00-17:00}}**: May to September, with: Monday to Friday 09:00-18:00, and Saturday to Sunday 13:00-17:00.
- **may{1-31}**: All of May.
- **may,september{1-15}**: All of May, and only September 1-15.
- **may,september{1-15{monday-friday}}**: All of May. And only September 1-15 Monday to Friday.
A BNF grammar for allowed times is given in section 3.2.1 of the *Developer Manual*.

The following action scripts have some additional options:

- **Send e-mail to administrators**: Additional options here are:
  - `info`: body of text inserted into the default e-mail message text, before the line beginning “Please take action”. The default text can be managed in the file `/cm/local/apps/cmd/scripts/actions/sendemail.py`
  - `recipients`: a list of recipients
  - `all admins`: uses the list of users in the Administrator e-mail setting in partition[base] mode

- **killprocess**, and **testaction**: Additional options for these are:
  - `arguments`: text that can be used by the script.
  - `script`: The location of the script on the file system.

### 12.4.5 Monitoring Configuration: Triggers

Triggers are introduced in section 12.2.5. The *Triggers* window (figure 12.18) allows actions (section 12.2.6) to be triggered based on conditions defined by the cluster administrator.

![Figure 12.18: Bright View Monitoring Configuration: Triggers](image)

**Change Detection For Triggers**

Triggers launch actions by detecting changes in the data of configured measurables. The detection of these changes can happen:

- **When a threshold is crossed**: That is: the latest sample value means that either the value has entered a zone when it was not in the zone in the preceding sample, or the latest sample means that the value has left a zone when it was in the zone in the preceding sample.
When the zone remains crossed. That is: the latest sample as well as the preceding sample are both within the zone of a crossed threshold.

When state flapping is detected. This is when the threshold is crossed repeatedly (5 times by default) within a certain time period (5 minutes by default).

The monitoring configuration dialog triggers have four possible action launch configuration options to cover these cases:

1. Enter actions: if the sample has entered into the zone and the previous sample was not in the zone. This is a threshold-crossing change.

2. Leave actions: if the sample has left the zone and the previous sample was in the zone. This is also a threshold-crossing change.

3. During actions: if the sample is in the zone, and the previous sample was also in the zone.

4. State flapping actions: if the sample is entering and leaving the zone within a particular period (State flapping period, 5 minutes by default) a set number of times (State flapping count, 5 by default).

Pre-defined Triggers: Passing, Failing, And Unknown Health Checks

By default, the only triggers that are pre-defined are the following three health check triggers, which use the Enter actions launch configuration option, and which have the following default behavior:

• Failing health checks: If a health check fails, then on entering the state of the health check failing, an event is triggered as the action, and a severity of 15 is set for that health check.

• Passing health checks: If a health check passes, then on entering the state of the health check passing, an event is triggered as the action, and a severity of 0 is set for that health check.

• Unknown health checks: If a health check has an unknown response, then on entering the state of the health check returning an unknown response, an event is triggered as the action, and a severity of 10 is set for that health check.

Example: carry out a triggered action: cmsh or Bright View can be used for the configuration of carrying out an e-mail alert action (that is: sending out an e-mail) that is triggered by failing health checks.

• A cmsh way to configure it is:

The e-mail action (send e-mail to administrators) is first configured so that the right recipients get a useful e-mail.

```
[root@bright82 ~]# cmsh
[bright82]% monitoring action
[bright82->monitoring->action]% use send e-mail to administrators
[bright82->...[send e-mail to administrators]]% append recipients user1@example.com
[bright82->...*[send e-mail to administrators*]]% commit
```

Here, email alerts would go to user1@example.com, as well as to anyone already configured in administratore-mail. Additional text can be set in the body of the e-mail by setting a value for info.

The trigger can be configured to run the action when the health check enters a state where its value is true: © Bright Computing, Inc.
[bright82->monitoring->action[use send e-mail to administrators]] % monitoring trigger
[bright82->monitoring->trigger] % use failing health checks
[bright82->monitoring->trigger[Failing health checks]] % append enteractions send e-mail to administrators
[bright82->monitoring->trigger*[Failing health checks*]] % commit

The settings can be viewed with the show command. TAB-completion prompting can be used to suggest possible values for the settings.

- A Bright View way to carry out the configuration is using the clickpath:
  Monitoring → Actions → Send e-mail to administrators → Edit
  This can be used to set the recipients and other items, and the configuration saved.
  The email action can then be configured in Bright View via the clickpath:
  Monitoring → Triggers → Failing Health Checks → Edit → Enter Actions ↓ Send E-mail to Administrators
  The checkbox for the “Send E-mail to Administrators” action should be ticked and the configuration saved.

**Adding Custom Triggers: Any Measurable, Any Action**

More triggers can be added. The `killallyestriigger` example from the basic example of section 12.1, seen in figures 12.3 and 12.4, is one such example.

The idea is that actions are launched from triggers, and the action for the trigger can be set to a predefined action, or to a custom action.

**The Expression Subwindow For A Trigger**

Clicking on the expression option in Trigger window of figure 12.18 opens up the expression subwindow. The expression for the trigger can then be configured by setting the entity, measurable, parameter, comparison operator, and measurable value, as shown in figure 12.19:

![Figure 12.19: Bright View Monitoring Configuration: Triggers Expression](image)

The trigger launch is carried out when, during sampling, CMDaemon evaluates the expression as being true.

© Bright Computing, Inc.
An example `cmsh` session to set up an expression for a custom trigger might be as follows, where the administrator is setting up the configuration so that an e-mail is sent by the monitoring system when a node is detected as having gone down:

**Example**

```
[root@bright82 ~]# cmsh
[bright82]% monitoring trigger add nodedown
[bright82->monitoring->trigger[nodedown]]% expression
[bright82->monitoring->trigger[nodedown]->expression[compare]]% show
Parameter Value
--------------------------------- ------------------------------------------------
Name compare
Revision
Type MonitoringCompareExpression
Entities Measurables
Parameters
Operator ==
Value FAIL
Use raw no
[bright82->monitoring->trigger[nodedown]->expression[compare]]% set value down
[bright82->monitoring->trigger[nodedown]->expression[compare]]% set operator eq
[bright82->monitoring->trigger[nodedown]->expression[compare]]% set measurables devicestate
```

To add a touch of realism, a deliberate mistake is set here—the use of `devicestate` (the data producer) instead of `devicestatus` (the measurable). The validate command (page 31) gives a helpful warning here, so that the cluster administrator can fix the setting:

```
[bright82->monitoring->trigger[nodedown]->expression[compare]]% validate
Field Message
----------------------------------------------------------------------------------------------
actions Warning: No actions were set
measurables/ Warning: No known measurable matches the specified regexes ('devicestate', '')
parameters
```

```
[bright82->monitoring->trigger[nodedown]->expression[compare]]% set measurables devicestatus
[bright82->monitoring->trigger[nodedown]->expression[compare]]% ..;
[bright82->monitoring->trigger[nodedown]]% set enteractions send e-mail to administrators
[bright82->monitoring->trigger[nodedown]]% commit
```

12.4.6 Monitoring Configuration: Health status

The Health status window (figure 12.20) displays all the nodes, and summarizes the results of all the health checks that have been run against them over time, by presenting a table of the associated severity levels (section 12.2.7):
12.4 Monitoring Configuration With Bright View

In the example shown in figure 12.20 the first entity shows a severity issue, while the other devices are fine. Details of the individual health checks per node can be viewed in a subwindow using the Show button for a node.

Clicking on the Show button for the first entity in this example opens up a subwindow (figure 12.21). For this example the issue turns out to be due to an UNKNOWN status in the ssh2node measurable.

12.4.7 Monitoring Configuration: All Health Checks

© Bright Computing, Inc.
The All Health checks window shows all the running health checks for all entities. If the filtering-by-column option at the top of the column for the node is used to show the results for that node only, then the results are similar to what the Show button for a node produces in section 12.4.6, figure 12.21.

12.4.8 Monitoring Configuration: Standalone Monitored Entities

The Standalone Monitored Entities window allows the cluster administrator to define a standalone entity. A standalone entity is one that is not managed by Bright Cluster Manager—which means that no CMDaemon is running on it to gather data and for managing it—but the entity can still be monitored. For example, a workstation that is running the Bright View browser could be the standalone entity. This could have its connectivity monitored by pinging it from the head node with a custom script.

12.5 The monitoring Mode Of cmsh

This section covers how to use cmsh to configure monitoring. The monitoring mode in cmsh corresponds generally to the Monitoring Configuration resource of Bright View in section 12.4. Similarly to how monitoring subwindows are accessed in Bright View, the monitoring mode of cmsh is itself is not used directly, except as a way to access the monitoring configuration submodes of cmsh.

For this section some familiarity is assumed with handling of objects as described in the introduction to working with objects (section 2.5.3). When using cmsh’s monitoring mode, the properties of objects in the submodes are how monitoring settings are carried out.

The monitoring mode of cmsh gives access to 8 modes under it:

Example

```
[root@myheadnode ~]# cmsh
[myheadnode]# monitoring help | tail -8
================================================= Monitoring ===============================
action ......................... Enter action mode
consolidator .................. Enter consolidator mode
labeledentity ................. Enter labeled entity mode
measurable ..................... Enter measurable mode
query.......................... Enter monitoring query mode
setup .......................... Enter monitoring configuration setup mode
standalone ..................... Enter standalone entity mode
trigger ........................ Enter trigger mode
```

For convenience, a tree for monitoring submodes is shown in figure 12.23.
12.5 The monitoring Mode Of cmsh

Sections 12.5.1–12.5.6 give examples of how objects are handled under these monitoring modes. To avoid repeating similar descriptions, section 12.5.1 is relatively detailed, and is often referred to by the other sections.

12.5.1 The action Submode

The action submode under the monitoring mode of cmsh allows monitoring actions to be configured. This mode in cmsh corresponds to the Bright View clickpath:

Monitoring Configuration—>Actions

described earlier in section 12.4.4:

The action mode handles action objects in the way described in the introduction to working with objects (section 2.5.3). A typical reason to handle action objects—the properties associated with an action script or action built-in—might be to view the actions available, or to add a custom action for use by, for example, a metric or health check.

Some examples of how the action mode is used are now give.

The action Submode: list, show, And get

The list command by default lists the names and properties of actions available from action mode in a table:

Example

© Bright Computing, Inc.
Cluster Monitoring

The preceding shows the actions available on a newly installed system.

The show command of cmsh displays the individual parameters and values of a specified action:

Example

```
[myheadnode->monitoring->action]% show poweroff
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Action</td>
<td>Power off a device</td>
</tr>
<tr>
<td>Allowed time</td>
<td></td>
</tr>
<tr>
<td>Disable</td>
<td>no</td>
</tr>
<tr>
<td>Name</td>
<td>PowerOff</td>
</tr>
<tr>
<td>Revision</td>
<td></td>
</tr>
<tr>
<td>Run on</td>
<td>Active</td>
</tr>
<tr>
<td>Type</td>
<td>PowerOff</td>
</tr>
</tbody>
</table>

Instead of using list, a convenient way to view the possible actions is to use the show command with tab-completion suggestions:

Example

```
[myheadnode->monitoring->action]% show<TAB><TAB>
```

drain  killprocess  powerreset  send \ e-mail \ to \ administrators  undrain
event  poweroff  reboot  shutdown
imageupdate  poweron  remount  testaction

The get command returns the value of an individual parameter of the action object:

Example

```
[myheadnode->monitoring->action]% get poweroff runon active
```

The action Submode: add, use, remove, commit, refresh, modified, set, clear, And validate

In the basic example of section 12.1, in section 12.1.2, the killallyes action was cloned from a similar script using a clone option in Bright View.

The equivalent can be done with a clone command in cmsh. However, using the add command instead, while it requires more steps, makes it clearer what is going on. This section therefore covers adding the killallyes script of section 12.1.2 using the add command.
When `add` is used: an object is added, the object is made the current object, and the name of the object is set, all at the same time. After that, `set` can be used to set values for the parameters within the object, such as a path for the value of the `parameter` command.

Adding an action requires that the type of action be defined. Just as tab-completion with `show` comes up with action suggestions, in the same way, using tab-completion with `add` comes up with type suggestions.

Running the command `help add` in the `action` mode also lists the possible types. These types are `drain`, `e-mail`, `event`, `imageupdate`, `poweroff`, `poweron`, `powerreset`, `reboot`, `script`, `servicerestart`, `servicestart`, `servicestop`, `shutdown`, `undrain`.

The syntax for the `add` command takes the form:

```
add <type> <action>
```

If there is no `killallyes` action already, then the name is added in the `action` mode with the `add` command, and the `script` type, as follows:

**Example**

```
[myheadnode->monitoring->action]% add script killallyes
[myheadnode->monitoring->action][killallyes]%
```

Using the `add` command drops the administrator into the `killallyes` object level, where its properties can be set. A successful commit means that the action is stored in CMDaemon.

The converse to the `add` command is the `remove` command, which removes an action that has had the `commit` command successfully run on it.

The `refresh` command can be run from outside the object level, and it removes the action if it has not yet been committed.

The `use` command is the usual way of "using" an object, where "using" means that the object being used is referred to by default by any command run. So if the `killallyes` object already exists, then `use` `killallyes` drops into the context of an already existing object (i.e. it "uses" the object).

The `set` command sets the value of each individual parameter displayed by a `show` command for that action. The individual parameter `script` can thus be set to the path of the `killallyes` script:

**Example**

```
[...oring->action][killallyes]% set script /cm/local/apps/cmd/scripts/actions/killallyes
```

The `clear` command can be used to clear the value that has been set for `script`.

The `validate` command checks if the object has all required values set to sensible values. So, for example, `commit` only succeeds if the `killallyes` object passes validation.

Validation does not check if the script itself exists. It only does a sanity check on the values of the parameters of the object, which is another matter. If the `killallyes` script does not yet exist in the location given by the parameter, it can be created as suggested in the basic example of section 12.1, in section 12.1.2. In the basic example used in this chapter, the script is run only on the head node. If it were to run on regular nodes, then the script should be copied into the disk image.

The `modified` command lists changes that have not yet been committed.

### 12.5.2 The consolidator Submode

Consolidators are introduced in section 12.4.3. Consolidators can be managed in `cmsh` via the `consolidator` mode, which is the equivalent of the consolidators window (section 12.4.3) in Bright View.

The `consolidator` mode deals with groups of consolidators. One such pre-defined group is `default`, while the other is `none`, as discussed earlier in section 12.4.3:
Each consolidators entry can have its parameters accessed and adjusted. For example, the parameters can be viewed with:

**Example**

```
[bright82->monitoring->consolidator]% use default
[bright82->monitoring->consolidator[default]]% show
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Revision</td>
<td></td>
</tr>
<tr>
<td>Kind</td>
<td>AVERAGE</td>
</tr>
<tr>
<td>Maximal age</td>
<td>0s</td>
</tr>
<tr>
<td>Maximal samples</td>
<td>4096</td>
</tr>
<tr>
<td>Name</td>
<td>day</td>
</tr>
<tr>
<td>Offset</td>
<td>0s</td>
</tr>
</tbody>
</table>

```
[bright82->monitoring->consolidator[default]->consolidators% use day
[bright82->monitoring->consolidator[default]->consolidators[day]]% show
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Interval</td>
<td>1d</td>
</tr>
</tbody>
</table>

Previously consolidated data is discarded with this type of change, if the number of samples is reduced. Changing parameters should therefore be done with some care.

A new consolidators group can be created if needed.

A Bright View way, where a decimalminutes group is created, is discussed in the example in section 12.4.3, page 481.

A cmsh way, where a max-per-day group is created, is discussed in the following section:

**Creation Of A Consolidator In cmsh**

A new consolidator group, max-per-day, can be added to the default consolidator groups of default and none, with:
12.5 The monitoring Mode Of cmsh

Example

[bright82]$ monitoring consolidator
[bright82->monitoring->consolidator]$ add max-per-day
[...[max-per-day*]]%

Within this new group, a new consolidator item, max-per-day can also be defined. The item can be
defined so that it only calculates the maximum value per day, using the kind setting. Another setting is
interval, which defines the interval with which the old data is compressed:

Example

[...[max-per-day*]]% consolidators
[...[max-per-day*]->consolidators]% add max-per-day
[...[max-per-day*]]% set interval 1d
[...[max-per-day*]]% set kind maximum
[...[max-per-day*]]% show
Parameter         Value
------------------- -----------------------------
Interval          1d
Kind              maximum
Maximal age       0s
Maximal samples   4096
Name              max-per-day
Offset            0s
Revision
[...[max-per-day*]]% commit

12.5.3 The measurable Submode
The measurable submode under the monitoring mode of cmsh handles measurable objects, that is:
metrics, health checks, and enummetrics. This mode corresponds to the Bright View clickpath:

Monitoring Configuration→Measurables

covered earlier in section 12.4.2.

Measurable objects represent the configuration of scripts or built-ins. The properties of the objects
are handled in cmsh in the way described in the introduction to working with objects (section 2.5.3).

A typical reason to handle measurable objects might be to view the measurables already available,
or to remove a measurable that is in use by an entity.

Measurables cannot be added from this mode. To add a measurable, its associated data producer
must be added from monitoring setup mode (section 12.5.4).

This section goes through a cmsh session giving some examples of how this mode is used.

The measurable Submode: list, show, And get
In measurable mode, the list command by default lists the names of all measurable objects along with
parameters, their class, and data producer.

Example

[bright82->monitoring->measurable]$ list
type   name (key) parameter class          producer
----------------------- ----------------- ----------------------------
Enum     DeviceStatus Internal          DeviceState
HealthCheck ManagedServicesOk Internal CMDaemonState
HealthCheck Mon::Storage Internal/Monitoring/Storage MonitoringSystem

© Bright Computing, Inc.
The above example illustrates a list with some of the measurables that can be set for sampling on a newly installed system. A full list typically contains over two hundred items.

The list command in measurable submode can be run as:

- list metric: to display only metrics
- list healthcheck: to display only health checks
- list enum: to display only enummetrics

The show command of the measurable submode of monitoring mode displays the parameters and values of a specified measurable, such as, for example CPUUser, devicestatus, or diskspace:

Example

Example

```
[myheadnode->monitoring->measurable]% show cpuuser
Parameter            Value
--------------------- ----------------------------
Class                CPU
Consolidator         default (ProcStat)
Cumulative           yes
Description          CPU time spent in user mode
Disabled             no (ProcStat)
Gap                   0 (ProcStat)
Maximal age          0s (ProcStat)
Maximal samples      4,096 (ProcStat)
Maximum              0
Minimum              0
Name                  CPUUser
Parameter             ProcStat
Revision              
Type                  Metric
Unit                  Jiffies
```

```
[myheadnode->monitoring->measurable]% show devicestatus
Parameter            Value
--------------------- ----------------------------
Class                Internal
Consolidator         none
Description          The device status
Disabled             no (DeviceState)
Gap                   0 (DeviceState)
Maximal age          0s (DeviceState)
Maximal samples      4,096 (DeviceState)
Name                  DeviceStatus
Parameter             DeviceState
Revision              
Type                  Enum
```

```
[myheadnode->monitoring->measurable]% show diskspace
Parameter            Value
--------------------- ----------------------------
```
The Gap setting here is a number. It sets how many samples are allowed to be missed before a value of NaN is set for the value of the metric.

As detailed in section 12.5.1, tab-completion suggestions for the show command suggest the names of objects that can be used, with the use command in this mode. For show in measurable mode, tab-completion suggestions suggests over 200 possible objects:

Example

[bright82->monitoring->measurable]% show
Display all 221 possibilities? (y or n)
alertlevel:count iotime:vda mon::storage::engine::elements oomkiller
alertlevel:maximum iotime:vdb mon::storage::engine::size opalinkhealth
alertlevel:sum ipforwdatagrams mon::storage::engine::usage packetsrecv:eth0
blockedprocesses ipfragcreates mon::storage::message::elements packetsrecv:eth1
buffermemory ipfragfails mon::storage::message::size packetssent:eth0
bytesrecv:eth0 ipfragoks mon::storage::message::usage packetssent:eth1
...

The single colon (":") indicates an extra parameter for that measurable.

Because there are a large number of metrics, it means that grepping a metrics list is sometimes handy. When listing and grepping, it is usually a good idea to allow for case, and be aware of the existence of the parameter column. For example, the AlertLevel metric shown in the first lines of the tab-completion suggestions of the show command of the previous example, shows up as alertlevel. However the list command displays it as AlertLevel. There are also several parameters associated with the AlertLevel command. So using the case-insensitive -i option of grep, and using the head command to display the headers is handy:

Example

[bright82->monitoring->measurable]% list | head -2 ; list metric | grep -i alertlevel
type name (key) parameter class producer
------------- -------------------- ---------- ------------------- ---------------
Metric AlertLevel count Internal AlertLevel
Metric AlertLevel maximum Internal AlertLevel
Metric AlertLevel sum Internal AlertLevel

The get command returns the value of an individual parameter of a particular health check object:

Example

[myheadnode->monitoring->measurable]% get oomkiller description
Checks whether oomkiller has come into action (then this check returns FAIL)
[myheadnode->monitoring->measurable]%
The **measurable Submode: The has Command**

The has command is used with a measurable to list the entities that use the measurable. Typically these are nodes, but it can also be other entities, such as the base partition.

**Example**

```markdown
[bright82->monitoring->measurable] has alertlevel:sum
bright82
node001
node002
```

The remaining commands in measurable mode, such as use, remove, commit, refresh, modified, set, clear, and validate; all work as outlined in the introduction to working with objects (section 2.5.3). More detailed usage examples of these commands within a monitoring mode are given in the earlier section covering the action submode (section 12.5.1).

**The measurable Submode: An Example Session On Viewing And Configuring A Measurable**

A typical reason to look at metrics and health check objects—the properties associated with the script or built-in—might be, for example, to view the operating sampling configuration for an entity.

This section goes through a cmsh example session under monitoring mode, where the setup submode (page 499) is used to set up a health check. The healthcheck can then be viewed from the measurable submode.

In the basic example of section 12.1, a trigger was set up from Bright View to check if the CPUUser metric was above 50 jiffies, and if so, to launch an action.

A functionally equivalent task can be set up by creating and configuring a health check, because metrics and health checks are so similar in concept. This is done here to illustrate how cmsh can be used to do something similar to what was done with Bright View in the basic example. A start is made on the task by creating a health check data producer, and configuring its measurable properties. using the setup mode under the monitoring mode of cmsh. The task is completed in the section on the setup mode in section 12.5.4.

To start the task, cmsh’s add command is used, and the type is specified, to create the new object:

**Example**

```markdown
[root@myheadnode ~]# cmsh

[myheadnode] monitoring setup
[myheadnode->monitoring->setup] add healthcheck cpucheck

[myheadnode->monitoring->setup*[cpucheck*]]%
```

The show command shows the parameters.

The values for description, runinbash, script, and class should be set:

**Example**

```markdown
[...->setup*[cpucheck*]]% set script /cm/local/apps/cmd/scripts/healthchecks/cpucheck
[...->setup*[cpucheck*]]% set description "CPUUser under 50%?"
[...->setup*[cpucheck*]]% set runinbash yes
[...->setup*[cpucheck*]]% set class OS
[...->setup*[cpucheck*]]% commit

[myheadnode->monitoring->setup[cpucheck]]%
```

On running commit, the data producer cpucheck is created:
Example

```
[myheadnode->monitoring->setup[cpucheck]]% exit; exit
[myheadnode->monitoring]% setup list | grep -i cpucheck
HealthCheckScript cpucheck 1 / 222 <0 in submode>
```

The measurable submode shows that a measurable cpucheck is also created:

Example

```
[myheadnode->monitoring]% measurable list | grep -i cpucheck
HealthCheck cpucheck OS cpucheck
```

Since the cpucheck script does not yet exist in the location given by the parameter script, it needs to be created. One ugly bash script that can do a health check is:

```
#!/bin/bash
## echo PASS if CPUUser < 50
## cpu is a %, ie: between 0 and 100

cpu='mpstat 1 1 | tail -1 | awk '{print $3}''
comparisonstring="$cpu" < 50"
if ((${bc <<< "$comparisonstring")}); then
echo PASS
else
  echo FAIL
fi
```

The script should be placed in the location suggested by the object, /cm/local/apps/cmd/scripts/healthchecks/cpucheck, and made executable with a chmod 700.

The cpucheck object is handled further within the cmsh monitoring setup mode in section 12.5.4 to produce a fully configured health check.

12.5.4 The setup Submode

The setup Submode: Introduction

The setup submode under the monitoring mode of cmsh allows access to all the data producers. This mode in cmsh corresponds to the Bright View clickpath:

Monitoring Configuration → Data Producers

covered earlier in section 12.4.1.

The setup Submode: Data Producers And Their Associated Measurables

The list of data producers in setup mode should not be confused with the list of measurables in measurable mode. Data producers are not the same as measurables. Data producers produce measurables, although it is true that the measurables are often named the same as, or similar to, their data producer.

In cmsh, data producers are in the Name (key) column when the list command is run from the setup submode:

Example

```
[bright82->monitoring->setup]% list
Type Name (key) Arguments Measurables Node execution filters
```

© Bright Computing, Inc.
In the preceding example, the AlertLevel data producer has $3 / 222$ as the value for measurables. This means that this AlertLevel data producer provides 3 measurables out of the 222 configured measurables. They may be enabled or disabled, depending on whether the data producer is enabled or disabled, but they are provided in any case.

To clarify this point: if the list command is run from setup mode to list producers, then the producers that have configured measurables are the ones that have 1 or more as the numerator value in the Measurables column. Conversely, the data producers with 0 in the numerator of the Measurables column have no configured measurables, whether enabled or disabled, and are effectively just placeholders until the software for the data producers is installed.

So, comparing the list of producers in setup mode with the measurables in measurable mode:

**Example**

In measurable mode, the three AlertLevel measurables (the 3 out of 222) produced by the AlertLevel producer can be seen with:

```
[bright82->monitoring->measurable]% list | head -2; list | grep AlertLevel

<table>
<thead>
<tr>
<th>Type</th>
<th>Name (key)</th>
<th>Parameter</th>
<th>Class</th>
<th>Producer</th>
</tr>
</thead>
<tbody>
<tr>
<td>Metric</td>
<td>AlertLevel</td>
<td>count</td>
<td>Internal</td>
<td>AlertLevel</td>
</tr>
<tr>
<td>Metric</td>
<td>AlertLevel</td>
<td>maximum</td>
<td>Internal</td>
<td>AlertLevel</td>
</tr>
<tr>
<td>Metric</td>
<td>AlertLevel</td>
<td>sum</td>
<td>Internal</td>
<td>AlertLevel</td>
</tr>
</tbody>
</table>
```

On the other hand, in measurable mode, there are no measurables seen for BigDataTools (the 0 out of 222) produced by the BigDataTools producer, when running, for example: list | head -2; list | grep BigDataTools.

**The setup Submode: Listing Nodes That Use A Data Producer**

The nodes command can be used to list the nodes on which a data producer <data producer> runs. It is run in the setup submode level of the monitoring mode as:

```
nodes <data producer>
```

**Example**

```
[bright82->monitoring->setup]% list | head -2; list | grep mount

<table>
<thead>
<tr>
<th>Type</th>
<th>Name (key)</th>
<th>Arguments</th>
<th>Measurables</th>
<th>Node execution filters</th>
</tr>
</thead>
<tbody>
<tr>
<td>HealthCheckScript</td>
<td>mounts</td>
<td></td>
<td>1 / 229</td>
<td>&lt;0 in submode&gt;</td>
</tr>
</tbody>
</table>
```

**The setup Submode: Data Producers Properties**

Any data producer from the full list in setup mode can, if suitable, be used to provide a measurable for any entity.

An example is the data producer AlertLevel. Its properties can be seen using the show command:
Example

[bright82->monitoring->setup]% show alertlevel
Parameter Value
Automatic reinitialize yes
Consolidator default
Description Alert level as function of all trigger severities
Disabled no
Execution multiplexer <1 in submode>
Fuzzy offset 0
Gap 0
Interval 2m
Maximal age 0s
Maximal samples 4096
Measurables 3 / 222
Name AlertLevel
Node execution filters <1 in submode>
Notes <0 bytes>
Offset 1m
Only when idle no
Revision
Type AlertLevel
When Timed

These properties are described in section 12.4.1. Most of these properties are inherited by the meaurables associated with the data producer, which in the AlertLevel data producer case are alertlevel:count, alertlevel:maximum, and alertlevel:sum.

The setup Submode: Deeper Submodes
One level under the setup submode of monitoring mode are 3 further submodes (modes deeper than submodes are normally also just called submodes for convenience, rather than sub-submodes):

- nodeexecutionfilters
- executionmultiplexers
- jobmetricsettings

Node execution filters: A way to filter execution (restrict execution) of the data producer.
If no node execution filter is set for that data producer, then the data producer runs on all nodes of the cluster. Filters are of type node, category, overlay, resource, and lua. The type is set when the filter is created.

- The nodes command for listing the execution nodes
Running the nodes command for a data producer lists which nodes the execution of the data producer is run on.

Example

[myhost->monitoring->setup]% nodes procmeminfo
mon001..mon003,myhost,osd001,osd002,node001,node002
[myhost->monitoring->setup]% nodes ssh2node
myhost
[myhost->monitoring->setup]% nodes devicestate
myhost
[myhost->monitoring->setup]% foreach * ( get name; nodes) | paste - - | sort
AggregateNode myhost
AggregatePDU myhost
AlertLevel myhost
Cassandra Not used
ceph_global Not used
chrootprocess myhost
ClusterTotal myhost
CMDaemonState node001,node002,myhost
...

Most of the default data producers that are used by the cluster run on an active head node, and often on the regular nodes.

- **nodexecutionfilters to restrict data producer execution**

  The `<rogueprocess>` (page 721) data producer is one of the few that by default runs on a regular node. Restricting a data producer to run on a particular list of nodes can be carried out as follows on a cluster that is originally in its default state:

  **Example**

  ```
  [bright82->monitoring->setup[rogueprocess]]% nodexecutionfilters
  [bright82->monitoring->setup[rogueprocess]]% nodeexecutionfilters
  category lua node overlay resource type
  [...] tup[rogueprocess]-%nodeexecutionfilters% add node justthese
  [...] tup*[rogueprocess*]-%nodeexecutionfilters*[justthese*]%% show
  Parameter Value
  ----------------- -----------------
  Filter operation Include
  Name justthese
  Nodes
  Revision
  Type Node
  [...] tup*[rogueprocess*]-%nodeexecutionfilters*[justthese*]%% set nodes node001,node002
  [...] tup*[rogueprocess*]-%nodeexecutionfilters*[justthese*]%% show
  Parameter Value
  ----------------- -----------------
  Filter operation Include
  Name justthese
  Nodes node001,node002
  Revision
  Type Node
  [...] tup*[rogueprocess*]-%nodeexecutionfilters*[justthese*]%% commit
  
  This way, the rogueprocess health check runs on just those nodes (node001, node002), and none of the others.

- **Restricting a data producer execution to the head node—monitoring a process on the head node**

  Another example of data producer restriction is as follows: an administrator may wish to monitor the `slapd` process on the head node. In `cmsh`, a session to achieve this could be:

  ```
  [bright82->monitoring->setup]% add procpidstat slapd
  [bright82->monitoring->setup*[slapd*]]% set process slapd
  ```
12.5 The monitoring Mode Of cmsh

The newly-defined slapd metric can now have its output displayed or plotted just like any other metric:

```
[bright82->device[bright82]]% latestmetricdata | grep slapd
MemoryUsed    slapd Process 739 MiB 28.6s
SystemTime    slapd Process 2m 35s 28.6s
ThreadsUsed   slapd Process 50 28.6s
UserTime      slapd Process 1h 36m 28.6s
VirtualMemoryUsed slapd Process 4.77 GiB 28.6s
```

• Filtering a data producer by resource

A data producer can also be set up so that it is run on a particular list of nodes filtered by resource. The resources that are available to a node can be viewed using the command `monitoringresources` for that device:

**Example**

```
[bright82->device[bright82]]% monitoringresources
Active
CentOS7u5
Ethernet
OpenStack Compute
RDO
boot
...
```

An example of where running a node execution filter by resource is useful, is for data producers that are intended to run on the active head node. Most data producers that are used by the cluster run on an active head node (besides often running on the regular nodes too).

Thus, for example, the `cpucheck` health check from page 498 can be set to run on the active head node, by creating an arbitrary resource called `myactive`:

**Example**

```
[bright82->monitoring->setup[cpucheck]]% nodeexecutionfilters
 [...typ[cpucheck]->nodeexecutionfilters]% add resource "myactive"
 [...typ[cpucheck]->nodeexecutionfilters*[myactive*]]% show
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Filter</td>
<td>Include</td>
</tr>
<tr>
<td>Name</td>
<td>myactive</td>
</tr>
<tr>
<td>Operator</td>
<td>OR</td>
</tr>
<tr>
<td>Resources</td>
<td></td>
</tr>
<tr>
<td>Revision</td>
<td></td>
</tr>
<tr>
<td>Type</td>
<td>Resource</td>
</tr>
</tbody>
</table>

and then setting the Resources parameter to Active:

© Bright Computing, Inc.
The cpucheck health check then runs on the active head node, whichever head node it is.

When node execution filtering is carried out, the filtered data is not dropped by default. Filtered data can be dropped for a measurable or an entity with the `monitoringdrop` command (section 12.6.7).

**Execution multiplexer:** A way to multiplex execution (have execution work elsewhere) for a data producer. It tells Bright Cluster Manager about the entities that the data producer is sampling for. A data producer runs and gathers data at the entity (node, category, lua, overlay, resource type) defined by the node execution filter, and with multiplex execution the data producer gathers samples from other entities. These entities can be nodes, categories, lua scripts, overlays, resources, and types. The entities from which it can sample are defined into groups called execution multiplexers. Execution multiplexers can thus be node multiplexers, category multiplexers, lua multiplexers, type multiplexers, overlay multiplexers, or resource multiplexers.

The `executionmultiplexers` mode can be entered for a data producer `dmesg` with:

**Example**

```
root@bright82 ~]# cmsh
[bright82]% monitoring setup executionmultiplexers dmesg
```

Running the commands: `help add`, or `help set`, can be used to show the valid syntax in this submode.

Most data producers run on a head node, but sample from the regular nodes. So, for example, the `dmesg` health check from Appendix G.2.1 can be set to sample from the regular nodes by setting it to carry out execution multiplexing to specified node entities using a node multiplexer with the arbitrary name of `nodes` as follows:

**Example**

```
[bright82->monitoring->setup[dmesg]->executionmultiplexers]% add<TAB><TAB>
category lua node overlay resource type
[bright82->monitoring->setup[dmesg]->executionmultiplexers]% add node nodes
[bright82->...*[dmesg*]->executionmultiplexers*[nodes*]]% show
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Filter operation</td>
<td>Include</td>
</tr>
<tr>
<td>Name</td>
<td>nodes</td>
</tr>
<tr>
<td>Nodes</td>
<td></td>
</tr>
<tr>
<td>Revision</td>
<td></td>
</tr>
<tr>
<td>Type</td>
<td>Node</td>
</tr>
</tbody>
</table>

```
[bright82->...*[dmesg*]->executionmultiplexers*[nodes*]]% set nodes node001,node002
```

```
[bright82->...*[dmesg*]->executionmultiplexers*[nodes*]]% show
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>
12.5 The monitoring Mode Of cmsh

Filter operation: Include
Name: nodes
Nodes: node001,node002
Revision: Node

The concepts and expected behavior of node execution filters and execution multiplexers is covered in more explicit detail in Appendix L.

Job Metrics Settings: Job metrics settings are a submode for setting job metric collection options for the JobSampler data producer (section 12.8.3).

12.5.5 The standalone Submode

The standalone submode under the monitoring mode of cmsh allows entities that are not managed by Bright Cluster Manager to be configured for monitoring. This mode in cmsh corresponds to the Bright View clickpath:

Monitoring Configuration→Standalone Monitored Entities

covered earlier in section 12.4.8.

The monitoring for such entities has to avoid relying on a CMDaemon that is running on the entity. An example might be a chassis that is monitored via a ping script running on the Bright Cluster Manager head node.

12.5.6 The trigger Submode

The trigger submode under the monitoring mode of cmsh allows actions to be configured according to the result of a measurable.

This mode in cmsh corresponds to the Bright View clickpath:

Monitoring Configuration→Triggers

covered earlier in section 12.4.5.

By default, there are 3 triggers:

Example

[bright82->monitoring->trigger]% list
Name (key) Expression Enter actions During actions Leave actions
---------------------------------------- ------------- -------------- -------------
Failing health checks (*, *, *) == FAIL Event
Passing health checks (*, *, *) == PASS Event
Unknown health checks (*, *, *) == UNKNOWN Event

Thus, for a passing, failing, or unknown health check, an event action takes place if entering a state change. The default severity level of a passing health check does not affect the AlertLevel value. However, if the failing or unknown health checks are triggered on entering a state change, then these will affect the AlertLevel value.

The trigger Submode: Setting An Expression

In the basic example of section 12.1, a trigger to run the killallyes script was configured using Bright View.

The expression that was set for the killallyes script in the basic example using Bright View can also be set in cmsh. For example:

© Bright Computing, Inc.
Example

```
[bright82->monitoring->trigger]% add killallyestrigger
[bright82->monitoring->trigger*[killallyestrigger*]]% show
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Disabled</td>
<td>no</td>
</tr>
<tr>
<td>During actions</td>
<td></td>
</tr>
<tr>
<td>Enter actions</td>
<td></td>
</tr>
<tr>
<td>Leave actions</td>
<td></td>
</tr>
<tr>
<td>Mark entity as failed</td>
<td>yes</td>
</tr>
<tr>
<td>Mark entity as unknown</td>
<td>no</td>
</tr>
<tr>
<td>Name</td>
<td>killallyestrigger</td>
</tr>
<tr>
<td>Revision</td>
<td></td>
</tr>
<tr>
<td>Severity</td>
<td>10</td>
</tr>
<tr>
<td>State flapping actions</td>
<td></td>
</tr>
<tr>
<td>State flapping count</td>
<td>5</td>
</tr>
<tr>
<td>State flapping period</td>
<td>5m</td>
</tr>
<tr>
<td>expression</td>
<td></td>
</tr>
</tbody>
</table>

```
[bright82->monitoring->trigger*[killallyestrigger*]]% expression
[bright82->monitoring->trigger*[killallyestrigger*]->expression[]]% show
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Entities</td>
<td></td>
</tr>
<tr>
<td>Measurables</td>
<td></td>
</tr>
<tr>
<td>Operator</td>
<td>EQ</td>
</tr>
<tr>
<td>Name</td>
<td></td>
</tr>
<tr>
<td>Parameters</td>
<td>MonitoringCompareExpression</td>
</tr>
<tr>
<td>Revision</td>
<td></td>
</tr>
<tr>
<td>Type</td>
<td></td>
</tr>
<tr>
<td>Use raw</td>
<td>no</td>
</tr>
<tr>
<td>Value</td>
<td>FAIL</td>
</tr>
</tbody>
</table>

```
[bright82->monitoring->trigger*[killallyestrigger*]->expression[]]% set entities bright82
[bright82->monitoring->trigger*[killallyestrigger*]->expression[*]]% set measurables CPUUser
[bright82->monitoring->trigger*[killallyestrigger*]->expression[*]]% set operator GT
[bright82->monitoring->trigger*[killallyestrigger*]->expression[*]]% set value 50
[bright82->monitoring->trigger*[killallyestrigger*]->expression[*]]% commit
[bright82->monitoring->trigger*[killallyestrigger*]->expression[*]]% set name killallyesregex
```

Field Message
```
------------------------ -------------------------------------------------------
actions                Warning: No actions were set
Field                   Message
```

The expression format is shown in cmsh as:

```
(<entity>, <measurable>, <parameter>) <comparison operator> <value>
```

Here:

- an entity, as described in section 12.2.1, can be, for example, a node, category, device, or software image. To include more than one entity for the comparison, the alternation (pipe, |) symbol can be used, with double quotes to enclose the expression.
12.5 The monitoring Mode Of cmsh

Example

...[killallyestrigger*]->expression[]% set entities "bright82|node001|compute|gpuimage"

In the preceding example, the entity compute could be a category, and the entity gpuimage could be a software image.

• a measurable (section 12.2.1) can be a health check, a metric, or an enummetric. For example: 
  CPUUsage. Alternation works for <measurable> in a similar way to that for <entity>.

• a parameter is a further option to a measurable. For example, the FreeSpace metric can take a mount point as a parameter. Alternation works for <parameter> in a similar way to that for <entity>.

• the comparison operator can be:
  EQ: equivalent to, displayed as ==
  NE: not equivalent to, displayed as !=
  GT: greater than, displayed as >
  LT: less than, displayed as <

If the user uses an arithmetic symbol such as > in cmsh as an unescaped entry, then the entry may unintentionally be interpreted by the shell. That is why the two-letter entries are used instead for entry, even though when displayed they display like the arithmetic symbols for easier recognition.

• the value can be a string, or a number.

The regex evaluates to TRUE or FALSE. The trigger runs its associated action in the case of TRUE. The wildcard * implies any entity, measurable, or parameter when used with the appropriate position according to the syntax of the expression format.

Using .* is also possible to match zero or more of any characters.

Some further expression matching examples:

Example

True for any failing health check:

(*, *, *) == FAIL

Example

True for any nearly full local disk (less than 10MB left):

(*, FreeSpace, sd[a-z]) < 10MB

Example

True for any cloud node that is too expensive (price more than more than 10$):

(.,*cnode.*, Price, *) > 10$

Example

Excluding node agw001:

(~(?!*.*agw001).*\$, *, *) == FAIL

Example

© Bright Computing, Inc.
True for any node in the data, gpu, or hpc categories, that has a nearly full local disk (less than 10MB left):

(!resource=category:data|category:gpu|category:hpc, FreeSpace, sd[a-z]) < 10MB

The unusual syntax in the preceding example is liable to change in future versions.

At the end of section 12.5.3 a script called cpucheck was built. This script was part of a task to use health checks instead of metrics to set up the functional equivalent of the behavior of the basic example of section 12.1. In this section the task is continued and completed as follows:

```
[bright82->monitoring->trigger]% expression killallyestrigger
[...trigger[killallyestrigger]->expression[killallyesregex]]% get measurables
CPUUser
[...trigger[killallyestrigger]->expression[killallyesregex]]% set measurables cpucheck
[...trigger*[killallyestrigger*]->expression*[killallyesregex*]]% commit
```

### 12.6 Obtaining Monitoring Data Values

The monitoring data values that are logged by devices can be used to generate graphs using the methods in section 12.3. However, sometimes an administrator would like to have the data values that generate the graphs instead, perhaps to import them into a spreadsheet for further direct manipulation, or to pipe them into a utility such as gnuplot.

#### 12.6.1 Getting The List Of Measurables For An Entity: The `measurables`, `metrics`, `healthchecks` And `enummetrics` Commands

The measurables for a specified entity can be seen with the `measurables` command, and the measurable subtypes can be seen with the corresponding measurable subset commands: `metrics`, `healthchecks` and `enummetrics`. The results look quite similar to the results of the `measurable` submode of the `monitoring` mode (section 12.5.3). However, for entities, the measurables are a sublist of the full number of measurables listed in the `measurable` submode, which in turn are only the list of measurables for the data producers that have been enabled.

For example, within `device` mode where the entities are typically the head node and regular nodes, running `metrics` with a specified entity shows only the metrics that are configured for that entity. Thus if the entity is a head node, then only head node metrics are shown; and if the entity is a regular node, only regular node metrics are shown:

**Example**

```
[bright82->device]% enummetrics node001
Type Name Parameter Class Producer
---------------- ----- ------------- ------------- ---------------
Enum DeviceStatus Internal DeviceState

[bright82->device]% use bright82
[bright82->device[bright82]]% measurables
Type Name Parameter Class Producer
---------------- ----- ------------- ------------- ---------------
Enum DeviceStatus Internal DeviceState
HealthCheck ManagedServicesOk Internal/CMDaemonState
HealthCheck Mon::Storage Internal/Monitoring/Storage MonitoringSystem

[bright82->device[bright82]]% exit
[bright82->device]% metrics node001
Type Name Parameter Class Producer
---------------- ----- ------------- ------------- ---------------
```
12.6 Obtaining Monitoring Data Values

Typically the number of metrics listed on the head node will differ from those listed on a regular node. Whatever each number is, it cannot be more than the number of metrics seen in the number of metrics listed in the measurable submode of section 12.5.3.

The preceding example shows the measurables listing commands being carried out on head nodes and regular nodes. These commands can be used on other entities too. For example, the base partition in partition mode, where the measurables can be listed with:

Example

```bash
[bright82->device]# partition use base
[bright82->partition[base]]% measurables
```

<table>
<thead>
<tr>
<th>Type</th>
<th>Name</th>
<th>Parameter</th>
<th>Class</th>
<th>Producer</th>
</tr>
</thead>
<tbody>
<tr>
<td>Metric</td>
<td>CoresTotal</td>
<td>Total</td>
<td>ClusterTotal</td>
<td></td>
</tr>
<tr>
<td>Metric</td>
<td>CoresUp</td>
<td>Total</td>
<td>ClusterTotal</td>
<td></td>
</tr>
<tr>
<td>Metric</td>
<td>DevicesClosed</td>
<td>Total</td>
<td>ClusterTotal</td>
<td></td>
</tr>
<tr>
<td>Metric</td>
<td>DevicesDown</td>
<td>Total</td>
<td>ClusterTotal</td>
<td></td>
</tr>
</tbody>
</table>

The values for metric samples and health checks can be obtained from within device mode in various ways, and are explained next.

12.6.2 On-Demand Metric Sampling And Health Checks

The `samplenow` Command For On-Demand Measurable Samples

An administrator can do live sampling, or sampling on-demand, for specified entities by using the `samplenow` command. The command has the following syntax:

```
samplenow [OPTIONS] [<entity>] [<measurable> ...]
```

The command can be run without options when an entity object, such as a node is used (output truncated):

Example

```
[bright82->device]# use bright82
[bright82->device[bright82]]% samplenow
```

<table>
<thead>
<tr>
<th>Measurable</th>
<th>Parameter</th>
<th>Type</th>
<th>Value</th>
<th>Age</th>
<th>Info</th>
</tr>
</thead>
<tbody>
<tr>
<td>AlertLevel</td>
<td>count</td>
<td>Internal</td>
<td>0</td>
<td>2.01s</td>
<td></td>
</tr>
<tr>
<td>AlertLevel</td>
<td>maximum</td>
<td>Internal</td>
<td>0</td>
<td>2.01s</td>
<td></td>
</tr>
<tr>
<td>AlertLevel</td>
<td>sum</td>
<td>Internal</td>
<td>0</td>
<td>2.01s</td>
<td></td>
</tr>
<tr>
<td>BlockedProcesses</td>
<td>OS</td>
<td>0 processes</td>
<td>2.01s</td>
<td></td>
<td></td>
</tr>
<tr>
<td>BufferMemory</td>
<td>Memory</td>
<td>847 KiB</td>
<td>2.01s</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The entity used can also be in other modes that have measurables, such as the base partition (output truncated):

Example

© Bright Computing, Inc.
The -n|--nodes Option
The -n option is used to sample specified nodes or node ranges:

Example

[device]\% device
[device]\% samplenow -n node001..node002 loadone

Entity Measurable Parameter Type Value Age Info
------ ------------------ --------- --------- ------- ------- -----
node001 LoadOne OS 0.04 0.08s
node002 LoadOne OS 0 0.077s

The --metrics And --checks Option
For a particular entity:

• All metrics can be sampled on demand with the --metrics option

• All health checks can be sampled on demand with the --checks option

Example

[device]\% samplenow --metrics loadone loadfifteen --n node001,node002

Entity Measurable Parameter Type Value Age Info
------ ------------------ --------- --------- ------- ------- -----
node001 LoadOne OS 0.04 0.08s
node002 LoadOne OS 0 0.077s

[device]\% samplenow --checks -n node001..node002

Entity Measurable Parameter Type Value Age Info
------ ------------------ --------- --------- ------- ------- -----
node001 ManagedServicesOk Internal PASS 0.177s
node001 defaultgateway Network PASS 0.145s
node001 diskspac e Disk PASS 0.16s
node001 dmesg OS PASS 0.177s

The -s|--status Option
Nodes in device mode which have a status of UP, as seen by the status command, can be sampled with the -s|--status option:

Example
12.6 Obtaining Monitoring Data Values

```
[bright82->device]% samplenow -s UP
Entity Measurable Parameter Type Value Age Info
--------- ------------------ --------- --------- ----------- ------ ----- 
  bright82 AlertLevel count Internal 0 4.67s 
  bright82 AlertLevel maximum Internal 0 4.67s 
  bright82 AlertLevel sum Internal 0 4.67s 
  bright82 BlockedProcesses DS 0 processes 4.67s 
  bright82 BufferMemory Memory 847 KiB 4.67s 
  bright82 BytesRecv eth0 Network 357 MiB 4.67s 
  bright82 BytesRecv eth1 Network 78.7 MiB 4.67s 

... 
```

The preceding example is truncated because it is quite lengthy. However, on the screen, for the device mode, it shows all the sample values for the measurables for all the entities—head node and regular nodes—that are up.

To restrict the results to node001 only, it can be run as:

```
Example

[bright82->device]% samplenow -s UP -n node001
```

```
Measureable Parameter Type Value Age Info
------------- --------- --------- ------ ----- 
  AlertLevel count Internal 0 0.081s 
  AlertLevel maximum Internal 0 0.081s 
  AlertLevel sum Internal 0 0.081s 

... 
```

Sampling according to a device status value other than UP is also possible.

The help text for the samplenow command gives further details on its possible options.

The latestmetricdata and latesthealthdata commands (section 12.6.3) display the results from the latest metric and health samples that have been gathered by the cluster, rather than sampling on demand.

The dumpmonitoringdata command (section 12.6.4) displays monitoring data gathered over a period of time in a variety of formats.

### 12.6.3 The Latest Data And Counter Values—The latest*data And latestmetriccounters Commands

Within device mode, the values obtained by the latest measurable sampling run can be displayed for a specified entity with the latestmonitoringdata, latestmetricdata and latesthealthdata commands:

- latestmetricdata: The latestmetricdata command for a specified entity displays the most recent metric value that has been obtained by the monitoring system for each metric used by the entity. For displaying metrics on-demand in cmsh, the samplenow --metrics command (page 510) can be used for a specified entity.

- latesthealthdata: The latesthealthdata command for a specified entity displays the most recent value that has been obtained by the monitoring system for each health check used by the entity. For displaying health check responses on demand in cmsh, the samplenow --checks command (page 510) can be used for a specified entity.

- latestmonitoringdata: The latestmonitoringdata command for a specified entity combines the output of the latesthealthdata and latestmetricdata commands, i.e. it displays the latest samples of the measurables for that entity. For displaying measurables on-demand in cmsh, the samplenow command (page 509) can be run without options, for a specified entity.
The `latestmetriccounters` command, on the other hand, displays the latest cumulative counter values of the cumulative metrics in use by the entity.

### Using The `latestmetricdata` Commands

When using the `latestmetricdata` commands, the device must be specified (some output elided):

#### Example

```
[bright82->device]% use node001
[bright82->device[node001]]% latestmetricdata
```

<table>
<thead>
<tr>
<th>Measurable</th>
<th>Parameter</th>
<th>Type</th>
<th>Value</th>
<th>Age</th>
<th>Info</th>
</tr>
</thead>
<tbody>
<tr>
<td>AlertLevel count</td>
<td>Internal</td>
<td>0</td>
<td></td>
<td>1m 12s</td>
<td>FAIL schedulers</td>
</tr>
<tr>
<td>AlertLevel maximum</td>
<td>Internal</td>
<td>0</td>
<td></td>
<td>1m 12s</td>
<td>FAIL schedulers</td>
</tr>
<tr>
<td>AlertLevel sum</td>
<td>Internal</td>
<td>0</td>
<td></td>
<td>1m 12s</td>
<td></td>
</tr>
<tr>
<td>BlockedProcesses</td>
<td>OS</td>
<td>0</td>
<td>processes</td>
<td>1m 12s</td>
<td></td>
</tr>
<tr>
<td>BufferMemory</td>
<td>Memory</td>
<td>847 Kib</td>
<td></td>
<td>1m 12s</td>
<td></td>
</tr>
<tr>
<td>BytesRecv eth0</td>
<td>Network</td>
<td>311.611 B/s</td>
<td></td>
<td>1m 12s</td>
<td></td>
</tr>
<tr>
<td>BytesRecv eth1</td>
<td>Network</td>
<td>0 B/s</td>
<td></td>
<td>1m 12s</td>
<td></td>
</tr>
<tr>
<td>BytesSent eth0</td>
<td>Network</td>
<td>349.953 B/s</td>
<td></td>
<td>1m 12s</td>
<td></td>
</tr>
<tr>
<td>BytesSent eth1</td>
<td>Network</td>
<td>0 B/s</td>
<td></td>
<td>1m 12s</td>
<td></td>
</tr>
<tr>
<td>CPUGuest</td>
<td>CPU</td>
<td>0 Jiffies</td>
<td></td>
<td>1m 12s</td>
<td></td>
</tr>
</tbody>
</table>

Valid device grouping options and other options can be seen in the help text for the `latestmetricdata` and `latesthealthdata` commands.

#### Example

```
[bright82->device]% help latestmetricdata
```

**Name:** Latestmetricdata - Display the latest metric data

**Usage:** latestmetricdata [OPTIONS] [<entity>]

**Options:**
- `-v, --verbose`
  
  Be more verbose

  - `-n, --nodes <node>`
    
    List of nodes, e.g. node001..node015,node020..node028,node030
    
    or `/some/file/containing/hostnames`

  - `-g, --group <group>`
    
    Include all nodes that belong to the node group, e.g. testnodes
    
    or test01,test03

By default the data values are shown with human-friendly units. The `--raw` option displays the data values as raw units.

### Using The `latestmetriccounter` Command

The `latestmetriccounter` is quite similar to the `latestmetricdata` command, except that it displays only cumulative metrics, and displays their accumulated counts since boot. The `latestmonitoringcounter` command is an alias for this command.

#### Example
12.6 Obtaining Monitoring Data Values

The reader can compare the preceding example output against the example output of the latestmetricdata command (page 512) to become familiar with the meaning of cumulative output.

12.6.4 Data Values Over A Period—The dumpmonitoringdata Command

The dumpmonitoringdata command displays monitoring data values over a specified period. This is for an entity, such as a node in device mode, or the base partition in partition mode, or an image in softwareimage mode.

Using The dumpmonitoringdata Command

A concise overview of the dumpmonitoringdata command can be displayed by typing in “help dumpmonitoringdata” in a cmsh mode that has entities.

The usage of the dumpmonitoringdata command consists of the following options and mandatory arguments:

dumpmonitoringdata [OPTIONS] <start-time> <end-time> <measurable> [entity]

The mandatory arguments: The mandatory arguments for the times, the measurables being dumped, and the entities being sampled, have values that are specified as follows:

- The measurable <measurable> for which the data values are being gathered must always be given. Measurables currently in use can conveniently be listed by running the measurables command (section 12.6.1).
- If [entity] is not specified when running the dumpmonitoringdata command, then it must be set by specifying the entity object from its parent mode of cmsh (for example, with use node001 in device mode). If the mode is device mode, then the entity can also be specified via the options as a list, a group, an overlay, or a category of nodes.
- The time pair <start-time> or <end-time> can be specified as follows:
  - **Fixed time format**: The format for the times that make up the time pair can be:
    * [YY/MM/DD] HH:MM[SS]
      (If YY/MM/DD is used, then each time must be enclosed in double quotes)
    * The unix epoch time (seconds since 00:00:00 1 January 1970)
  - **now**: For the <end-time>, a value of now can be set. The time at which the dumpmonitoringdata command is run is then used.
  - **Relative time format**: One item in the time pair can be set to a fixed time format. The other item in the time pair can then have its time set relative to the fixed time item. The format for the non-fixed time item (the relative time item) can then be specified as follows:
    * For the <start-time>, a number prefixed with “-” is used. It indicates a time that much earlier to the fixed end time.
* For the <end-time>, a number prefixed with “+” is used. It indicates a time that much later to the fixed start time.
* The number values also have suffix values indicating the units of time, as seconds (s), minutes (m), hours (h), or days (d).

The relative time format is summarized in the following table:

<table>
<thead>
<tr>
<th>Unit</th>
<th>&lt;start-time&gt;</th>
<th>&lt;end-time&gt;</th>
</tr>
</thead>
<tbody>
<tr>
<td>seconds</td>
<td>-&lt;number&gt;s</td>
<td>+&lt;number&gt;s</td>
</tr>
<tr>
<td>minutes</td>
<td>-&lt;number&gt;m</td>
<td>+&lt;number&gt;m</td>
</tr>
<tr>
<td>hours</td>
<td>-&lt;number&gt;h</td>
<td>+&lt;number&gt;h</td>
</tr>
<tr>
<td>days</td>
<td>-&lt;number&gt;d</td>
<td>+&lt;number&gt;d</td>
</tr>
</tbody>
</table>

**The options:** The options applied to the samples are specified as follows:

<table>
<thead>
<tr>
<th>Option</th>
<th>Argument(s)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>-v, --verbose</td>
<td></td>
<td>show the rest of the line on a new line instead of cutting it off</td>
</tr>
<tr>
<td>-i, --intervals</td>
<td>&lt;number&gt;</td>
<td>number of samples to show</td>
</tr>
<tr>
<td>-u, --unix</td>
<td></td>
<td>use a unix timestamp instead of using the default date format</td>
</tr>
<tr>
<td>-d, --delimiter</td>
<td>&quot;&lt;string&gt;&quot;</td>
<td>set the delimiter to a character</td>
</tr>
<tr>
<td>--sum</td>
<td></td>
<td>sum over specified entities</td>
</tr>
<tr>
<td>--max</td>
<td></td>
<td>maximum over specified entities</td>
</tr>
<tr>
<td>--min</td>
<td></td>
<td>minimum over specified entities</td>
</tr>
<tr>
<td>--avg</td>
<td></td>
<td>average over specified entities</td>
</tr>
<tr>
<td>--raw</td>
<td></td>
<td>show the metric value without units</td>
</tr>
<tr>
<td>--human</td>
<td></td>
<td>show the metric value with units (default)</td>
</tr>
<tr>
<td>--consolidationinterval</td>
<td></td>
<td>retrieve data from the consolidator with specified interval</td>
</tr>
<tr>
<td>--consolidationoffset</td>
<td></td>
<td>retrieve data from the consolidator with specified (interval, offset)</td>
</tr>
<tr>
<td>--timeaverage</td>
<td></td>
<td>calculate the average for the entire interval for specified devices</td>
</tr>
<tr>
<td>--delta</td>
<td></td>
<td>display change relative to previous value</td>
</tr>
<tr>
<td>--clip</td>
<td></td>
<td>clip data samples to the requested interval</td>
</tr>
</tbody>
</table>

**The following options are valid only for device mode:**

<table>
<thead>
<tr>
<th>Option</th>
<th>Argument(s)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>-n, --nodes</td>
<td>&lt;list&gt;</td>
<td>for list of nodes</td>
</tr>
<tr>
<td>-g, --groups</td>
<td>&lt;list&gt;</td>
<td>for list of groups</td>
</tr>
<tr>
<td>-c, --categories</td>
<td>&lt;list&gt;</td>
<td>for list of categories</td>
</tr>
<tr>
<td>-r, --racks</td>
<td>&lt;list&gt;</td>
<td>for list of racks</td>
</tr>
<tr>
<td>-h, --chassis</td>
<td>&lt;list&gt;</td>
<td>for list of chassis</td>
</tr>
<tr>
<td>-e, --overlay</td>
<td>&lt;list&gt;</td>
<td>Include all nodes in list of overlays</td>
</tr>
<tr>
<td>--union</td>
<td></td>
<td>calculate the union of specified devices</td>
</tr>
<tr>
<td>--intersection</td>
<td></td>
<td>calculate the intersection of the specified devices</td>
</tr>
<tr>
<td>-l, --role</td>
<td>&lt;role&gt;</td>
<td>Filter all nodes in role</td>
</tr>
<tr>
<td>-s, --status</td>
<td>&lt;state&gt;</td>
<td>for nodes in state UP, OPENING, DOWN, and so on</td>
</tr>
</tbody>
</table>
Notes And Examples Of dumpmonitoringdata Command Use

Notes and examples of how the dumpmonitoringdata command can be used now follow:

Fixed time formats: Time pairs can be specified for fixed times:

Example

```
[bright82->device[node001]]% dumpmonitoringdata 18:00:00 18:02:00 loadone
```

<table>
<thead>
<tr>
<th>Timestamp</th>
<th>Value</th>
<th>Info</th>
</tr>
</thead>
<tbody>
<tr>
<td>2017/08/30 17:58:00</td>
<td>0.02</td>
<td></td>
</tr>
<tr>
<td>2017/08/30 18:00:00</td>
<td>0.01</td>
<td></td>
</tr>
<tr>
<td>2017/08/30 18:02:00</td>
<td>0.02</td>
<td></td>
</tr>
</tbody>
</table>

Double quotes are needed for times with a YY/MM/DD specification:

Example

```
[bright82->device[node001]]% dumpmonitoringdata "17/08/30 18:00" "17/08/30 18:02" loadone
```

<table>
<thead>
<tr>
<th>Timestamp</th>
<th>Value</th>
<th>Info</th>
</tr>
</thead>
<tbody>
<tr>
<td>2017/08/30 17:58:00</td>
<td>0.02</td>
<td></td>
</tr>
<tr>
<td>2017/08/30 18:00:00</td>
<td>0.01</td>
<td></td>
</tr>
<tr>
<td>2017/08/30 18:02:00</td>
<td>0.02</td>
<td></td>
</tr>
</tbody>
</table>

Unix epoch time can also be set:

Example

```
[bright82->device[node001]]% !date -d "Aug 30 18:00:00 2017" +%s
1504108800
[bright82->device[node001]]% dumpmonitoringdata 1504108800 1504108920 loadone
```

<table>
<thead>
<tr>
<th>Timestamp</th>
<th>Value</th>
<th>Info</th>
</tr>
</thead>
<tbody>
<tr>
<td>2017/08/30 17:58:00</td>
<td>0.02</td>
<td></td>
</tr>
<tr>
<td>2017/08/30 18:00:00</td>
<td>0.01</td>
<td></td>
</tr>
<tr>
<td>2017/08/30 18:02:00</td>
<td>0.02</td>
<td></td>
</tr>
</tbody>
</table>

Intervals and interpolation: The -i|--intervals option interpolates the data values that are to be displayed. The option needs <number> samples to be specified. This then becomes the number of interpolated samples across the given time range. Using “-i 0” outputs only the non-interpolated stored samples—the raw data—and is the default.

Example

```
[bright82->device[node001]]% dumpmonitoringdata -i 0 -10m now loadone node001
```

<table>
<thead>
<tr>
<th>Timestamp</th>
<th>Value</th>
<th>Info</th>
</tr>
</thead>
<tbody>
<tr>
<td>2017/07/21 14:56:00</td>
<td>0.01</td>
<td></td>
</tr>
<tr>
<td>2017/07/21 14:58:00</td>
<td>0.14</td>
<td></td>
</tr>
<tr>
<td>2017/07/21 15:00:00</td>
<td>0.04</td>
<td></td>
</tr>
<tr>
<td>2017/07/21 15:02:00</td>
<td>0.04</td>
<td></td>
</tr>
<tr>
<td>2017/07/21 15:04:00</td>
<td>0.08</td>
<td></td>
</tr>
<tr>
<td>2017/07/21 15:06:00</td>
<td>0.08</td>
<td></td>
</tr>
</tbody>
</table>

If the number of intervals is set to a non-zero value, then the last value is always no data, since it cannot be interpolated.
Example

[bright82->device]% dumpmonitoringdata -i 3 -10m now loadone node001

<table>
<thead>
<tr>
<th>Timestamp</th>
<th>Value</th>
<th>Info</th>
</tr>
</thead>
<tbody>
<tr>
<td>2017/07/21 21:49:36</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>2017/07/21 21:54:36</td>
<td>0.0419998</td>
<td></td>
</tr>
<tr>
<td>2017/07/21 21:59:36</td>
<td>no data</td>
<td></td>
</tr>
</tbody>
</table>

A set of nodes can be specified for the dump:

[bright82->device]% dumpmonitoringdata -n node001..node002 -5m now cpuidle

<table>
<thead>
<tr>
<th>Entity</th>
<th>Timestamp</th>
<th>Value</th>
<th>Info</th>
</tr>
</thead>
<tbody>
<tr>
<td>node001</td>
<td>2017/07/20 20:14:00</td>
<td>99.8258 Jiffies</td>
<td></td>
</tr>
<tr>
<td>node001</td>
<td>2017/07/20 20:16:00</td>
<td>99.8233 Jiffies</td>
<td></td>
</tr>
<tr>
<td>node001</td>
<td>2017/07/20 20:18:00</td>
<td>99.8192 Jiffies</td>
<td></td>
</tr>
<tr>
<td>node001</td>
<td>2017/07/20 20:20:00</td>
<td>99.8475 Jiffies</td>
<td></td>
</tr>
<tr>
<td>node002</td>
<td>2017/07/20 20:14:00</td>
<td>99.7917 Jiffies</td>
<td></td>
</tr>
<tr>
<td>node002</td>
<td>2017/07/20 20:16:00</td>
<td>99.8083 Jiffies</td>
<td></td>
</tr>
<tr>
<td>node002</td>
<td>2017/07/20 20:18:00</td>
<td>99.7992 Jiffies</td>
<td></td>
</tr>
<tr>
<td>node002</td>
<td>2017/07/20 20:20:00</td>
<td>99.815 Jiffies</td>
<td></td>
</tr>
</tbody>
</table>

Summing values: The -m|--sum option sums a specified metric for specified devices, for a set of specified times. For 2 nodes, over a period 2 hours ago, with values interpolated over 3 time intervals, the option can be used as follows:

Example

[bright82->device]% dumpmonitoringdata -2h now -i 3 loadone -n node00[1-2] --sum

<table>
<thead>
<tr>
<th>Timestamp</th>
<th>Value</th>
<th>Info</th>
</tr>
</thead>
<tbody>
<tr>
<td>2017/07/20 18:30:27</td>
<td>0.0292462</td>
<td></td>
</tr>
<tr>
<td>2017/07/20 19:30:27</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>2017/07/20 20:30:27</td>
<td>no data</td>
<td></td>
</tr>
</tbody>
</table>

Each entry in the values column in the preceding table is the sum of loadone displayed by node001, and by node002, at that time, as can be seen from the following corresponding table:

Example

[bright82->device]% dumpmonitoringdata -2h now -i 3 loadone -n node00[1-2]

<table>
<thead>
<tr>
<th>Entity</th>
<th>Timestamp</th>
<th>Value</th>
<th>Info</th>
</tr>
</thead>
<tbody>
<tr>
<td>node001</td>
<td>2017/07/20 18:30:27</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>node001</td>
<td>2017/07/20 19:30:27</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>node001</td>
<td>2017/07/20 20:30:27</td>
<td>no data</td>
<td></td>
</tr>
<tr>
<td>node002</td>
<td>2017/07/20 18:30:27</td>
<td>0.0292462</td>
<td></td>
</tr>
<tr>
<td>node002</td>
<td>2017/07/20 19:30:27</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>node002</td>
<td>2017/07/20 20:30:27</td>
<td>no data</td>
<td></td>
</tr>
</tbody>
</table>

Each loadone value shown by a node at a time shown in the preceding table, is in turn an average interpolated value, based on actual data values sampled for that node around that time.
Displaying values during a specified time period, with --clip: The --clip option is used with a specified time period. If there are raw values within the period, these are displayed.

A value is displayed for the start of the period, either by selection of a raw value if it exists at the exact starting time, or via interpolation if there is no raw value at the exact starting time. Similarly, at the end of the period a raw value is shown if it exists, or an interpolated value is shown if it does not.

Example

```
[bright82->device[node001]]% dumpmonitoringdata -5m now bytessent:eth0 --clip
# Start : 1552400295 / Tue Mar 12 14:18:15 2019
# End : 1552400595 / Tue Mar 12 14:23:15 2019
Timestamp Value Info
-------------------------- ------------ ----------
2019/03/12 14:18:15 201.942 B/s
2019/03/12 14:20:15 217.883 B/s
2019/03/12 14:22:15 233.058 B/s
2019/03/12 14:23:15 235.831 B/s
```

In the preceding example, the first 3 samples are raw samples, the last sample is an interpolated value, over a time period evaluated as being from 14:18:15 to 14:23:15. The epoch times for this period, and corresponding human-readable values are shown in the heading to the output table.

Displaying according to status: The -s|--status option selects only for nodes with the specified state. A state is one of the values output by the cmsh command ds or device status. It is also one of the values returned by the enummetric DeviceStatus (section 12.2.2).

Example

```
[bright82->device[node001]]% dumpmonitoringdata -2m now loadone -s up
Entity Timestamp Value Info
------------ -------------------------- ---------- ----------
bright82 2017/07/21 15:00:00 0.35
bright82 2017/07/21 15:02:00 0.53
node001 2017/07/21 14:12:00 0.04
node001 2017/07/21 15:02:00 0.04
node002 2017/07/21 15:00:00 0.22
node002 2017/07/21 15:02:00 0.21
```

The argument to -s|--status can be specified with simple regexes, which are case insensitive. For example, inst.* covers the states installing, installer_failed, installer_rebooting, installer_callinginit, installer_unreachable, installer_burning.

Displaying deltas: The --delta option lists the difference between successive monitoring data values. It subtracts the previous data value from the current data value, and divides the result by the time interval between the two values.

Example

```
[bright82->device[node001]]% dumpmonitoringdata --delta -6m now pageout
Timestamp Value Delta Info
-------------------------- ------------ --------------------------
2018/09/10 17:49:28 1015.46 B/s nan
2018/09/10 17:51:28 1.35 KiB/s 2.7 B/s/s
2018/09/10 17:53:28 1.34 KiB/s -0.083 B/s/s
```
Deltas are useful for seeing patterns in rates of change. For example, to check an experimental version of CMDaemon for a memory leak, an administrator may run:

**Example**

```bash
[bright82->device]% dumpmonitoringdata -2h now memoryused:cmd -n node001 --delta
```

<table>
<thead>
<tr>
<th>Timestamp</th>
<th>Value</th>
<th>Delta</th>
<th>Info</th>
</tr>
</thead>
<tbody>
<tr>
<td>2018/08/15 10:00:00.812</td>
<td>68.4 MiB</td>
<td>nan</td>
<td></td>
</tr>
<tr>
<td>2018/08/15 10:02:00.812</td>
<td>68.4 MiB</td>
<td>0.0341333 B/s</td>
<td></td>
</tr>
<tr>
<td>2018/08/15 12:10:00.812</td>
<td>68.4 MiB</td>
<td>0 B/s</td>
<td></td>
</tr>
</tbody>
</table>

The roughly 0B/s increase over 2 hours in the preceding output is a good sign.

**Displaying union and intersection sets:** The `--union` option displays the union of a set of specified devices. The devices can be specified by the device grouping options (the options that are used to group `<lists>`, such as `-c`, `-r` and so on).

For example:

- if the overlay galeranodes has the node mon001
- and
- the overlay openstackhypervisors has the nodes node001, and node002

then an example of a union of the set of these two overlays is:

**Example**

```bash
[bright82->device]% dumpmonitoringdata --union -3m now pageout -e galeranodes,openstackhypervisors
```

<table>
<thead>
<tr>
<th>Entity</th>
<th>Timestamp</th>
<th>Value</th>
<th>Info</th>
</tr>
</thead>
<tbody>
<tr>
<td>mon001</td>
<td>2018/09/11 11:31:56.198</td>
<td>192 KiB/s</td>
<td></td>
</tr>
<tr>
<td>mon001</td>
<td>2018/09/11 11:33:56.198</td>
<td>17.8 KiB/s</td>
<td></td>
</tr>
<tr>
<td>node002</td>
<td>2018/09/11 11:32:04.509</td>
<td>1.54 KiB/s</td>
<td></td>
</tr>
<tr>
<td>node002</td>
<td>2018/09/11 11:34:04.509</td>
<td>1.30 KiB/s</td>
<td></td>
</tr>
</tbody>
</table>

A union of sets in the same grouping option can be carried out using comma-separation for the list of sets. In the preceding example, the same grouping option is `-e|--overlay`.

For a union of different grouping options however, the syntax is different. For example, for a union of the galeranodes overlay, and a node001 node, a similar example is:

**Example**

```bash
[bright82->device]% dumpmonitoringdata --union -3m now pageout -u -e galeranodes -n node001
```

<table>
<thead>
<tr>
<th>Entity</th>
<th>Timestamp</th>
<th>Value</th>
<th>Info</th>
</tr>
</thead>
<tbody>
<tr>
<td>mon001</td>
<td>1536659036.198</td>
<td>17.3 KiB/s</td>
<td></td>
</tr>
<tr>
<td>mon001</td>
<td>1536659156.198</td>
<td>116 KiB/s</td>
<td></td>
</tr>
<tr>
<td>node001</td>
<td>1536659008.997</td>
<td>1023.99 B/s</td>
<td></td>
</tr>
<tr>
<td>node001</td>
<td>1536659128.996</td>
<td>1.26 KiB/s</td>
<td></td>
</tr>
</tbody>
</table>

For an intersection of sets, the only syntax allowed is one that uses different grouping options:

**Example**

```bash
[bright82->device]% dumpmonitoringdata --intersection -3m now pageout -e galeranodes -n node001
```

No remaining entities

For intersection, comma-separation within one grouping option is pointless, and is not supported.
Some non-interpolating RLE quirks: When a sample measurement is carried out, if the sample has the same value as the two preceding it in the records, then the “middle” sample is discarded from storage.

Thus, when viewing the sequence of output of non-interpolated samples, identical values do not exceed two entries one after the other. This is a common compression technique known as Run Length Encoding (RLE). It can have some implications in the output of the dumpmonitoringdata command.

Example

[bright82->device[node001]]% dumpmonitoringdata -10m now threadsused:cmd

<table>
<thead>
<tr>
<th>Timestamp</th>
<th>Value</th>
<th>Info</th>
</tr>
</thead>
<tbody>
<tr>
<td>2017/07/21 11:16:00</td>
<td>42</td>
<td></td>
</tr>
<tr>
<td>2017/07/21 11:20:00</td>
<td>42</td>
<td></td>
</tr>
<tr>
<td>2017/07/21 11:22:00</td>
<td>41</td>
<td></td>
</tr>
<tr>
<td>2017/07/21 11:24:00</td>
<td>42</td>
<td></td>
</tr>
<tr>
<td>2017/07/21 11:26:00</td>
<td>42</td>
<td></td>
</tr>
</tbody>
</table>

In the preceding example, data values for the number of threads used by CMDaemon are dumped for the last 10 minutes.

Because of RLE, the value entry around 11:18:00 in the preceding example is skipped. It also means that at most only 2 of the same values are seen sequentially in the Value column. This means that 42 is not the answer to everything.

For a non-interpolated value, the nearest value in the past, relative to the time of sampling, is used as the sample value for the time of sampling. This means that for non-interpolated values, some care may need to be taken due to another aspect of the RLE behavior: The time over which the samples are presented may not be what a naive administrator may expect when specifying the time range. For example, if the administrator specifies a 10 minute time range as follows:

Example

[bright82->softwareimage]% dumpmonitoringdata -10m now nodesup default-image

<table>
<thead>
<tr>
<th>Timestamp</th>
<th>Value</th>
<th>Info</th>
</tr>
</thead>
<tbody>
<tr>
<td>2017/07/13 16:43:00</td>
<td>2</td>
<td></td>
</tr>
<tr>
<td>2017/07/20 17:37:00</td>
<td>2</td>
<td></td>
</tr>
</tbody>
</table>

then here, because the dump is for non-interpolated values, it means that the nearest value in the past, relative to the time of sampling, is used as the sample value. For values that are unlikely to change much, it means that rather than 10 minutes as the time period within which the samples are taken, the time period can be much longer. Here it turns out to be about 7 days because the nodes happened to be booted then.

12.6.5 Monitoring Data Health Overview–The healthoverview Command

In figure 12.20, section 12.4.6, the Bright View clickpath

Monitoring→Health Status

showed an overview of the health status of all nodes.

The cmsh equivalent is the healthoverview command, which is run from within device mode. If run without using a device, then it provides a summary of the alert levels for all nodes.

The help text in cmsh explains the options for the healthoverview command. The command can be run with options to restrict the display to specified nodes, and also to display according to the sort order of the alert level values.
Example

[bright82->device]% healthoverview -n node00[1-3]

<table>
<thead>
<tr>
<th>Device</th>
<th>Sum</th>
<th>Maximum</th>
<th>Count</th>
<th>Age</th>
<th>Info</th>
</tr>
</thead>
<tbody>
<tr>
<td>node001</td>
<td>30</td>
<td>15</td>
<td>2</td>
<td>50.7s</td>
<td>hot, fan high</td>
</tr>
<tr>
<td>node002</td>
<td>30</td>
<td>15</td>
<td>2</td>
<td>50.7s</td>
<td>hot, fan high</td>
</tr>
<tr>
<td>node003</td>
<td>15</td>
<td>15</td>
<td>1</td>
<td>50.7s</td>
<td>hot</td>
</tr>
</tbody>
</table>

12.6.6 Monitoring Data About The Monitoring System—The monitoringinfo Command

The monitoringinfo command provides information for specified head nodes or regular nodes about the monitoring subsystem. The help text shows the options for the command. Besides options to specify the nodes, there are options to specify what monitoring information aspect is shown, such as storage, cache, or services.

Example

[bright82->device]% monitoringinfo -n node001

<table>
<thead>
<tr>
<th>Service</th>
<th>Queued</th>
<th>Handled</th>
<th>Cache miss</th>
<th>Stopped</th>
<th>Suspended</th>
<th>Last operation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mon::CacheGather</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>yes</td>
<td>no</td>
<td>-</td>
</tr>
<tr>
<td>Mon::DataProcessor</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>yes</td>
<td>no</td>
<td>-</td>
</tr>
<tr>
<td>Mon::DataTranslator</td>
<td>0</td>
<td>932,257</td>
<td>0</td>
<td>no</td>
<td>no</td>
<td>Mon Jul 24 11:34:00</td>
</tr>
<tr>
<td>Mon::EntityMeasurableCache</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>no</td>
<td>no</td>
<td>Thu Jul 13 16:39:52</td>
</tr>
<tr>
<td>Mon::MeasurableBroker</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>no</td>
<td>no</td>
<td>-</td>
</tr>
<tr>
<td>Mon::Replicate::Collector</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>yes</td>
<td>yes</td>
<td>-</td>
</tr>
<tr>
<td>Mon::Replicate::Combiner</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>yes</td>
<td>yes</td>
<td>-</td>
</tr>
<tr>
<td>Mon::RepositoryAllocator</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>yes</td>
<td>no</td>
<td>-</td>
</tr>
<tr>
<td>Mon::RepositoryTrim</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>yes</td>
<td>no</td>
<td>-</td>
</tr>
<tr>
<td>Mon::TaskInitializer</td>
<td>0</td>
<td>30</td>
<td>0</td>
<td>no</td>
<td>no</td>
<td>Thu Jul 13 16:39:52</td>
</tr>
<tr>
<td>Mon::TaskSampler</td>
<td>30</td>
<td>233,039</td>
<td>0</td>
<td>no</td>
<td>no</td>
<td>Mon Jul 24 11:34:00</td>
</tr>
<tr>
<td>Mon::Trigger::Actuator</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>yes</td>
<td>no</td>
<td>-</td>
</tr>
<tr>
<td>Mon::Trigger::Dispatcher</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>yes</td>
<td>no</td>
<td>-</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Cache</th>
<th>Size</th>
<th>Updates</th>
<th>Requests</th>
</tr>
</thead>
<tbody>
<tr>
<td>ConsolidatorCache</td>
<td>0</td>
<td>17</td>
<td>0</td>
</tr>
<tr>
<td>EntityCache</td>
<td>10</td>
<td>17</td>
<td>935,280</td>
</tr>
<tr>
<td>GlobalLastRawDataCache</td>
<td>87</td>
<td>17</td>
<td>0</td>
</tr>
<tr>
<td>LastRawDataCache</td>
<td>142</td>
<td>17</td>
<td>427,301</td>
</tr>
<tr>
<td>MeasurableCache</td>
<td>231</td>
<td>17</td>
<td>935,230</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Cache</th>
<th>Up</th>
<th>Down</th>
<th>Closed</th>
</tr>
</thead>
<tbody>
<tr>
<td>DeviceStateCache</td>
<td>3</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Replicator</th>
<th>First</th>
<th>Last</th>
<th>Requests</th>
<th>Samples</th>
<th>Sources</th>
</tr>
</thead>
<tbody>
<tr>
<td>ReplicateRequestHandler</td>
<td>-</td>
<td>-</td>
<td>0</td>
<td>0</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Cache</th>
<th>Queued</th>
<th>Delivered</th>
<th>Handled</th>
<th>Pickup</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cache</td>
<td>0</td>
<td>120</td>
<td>932,257</td>
<td>7,766</td>
</tr>
</tbody>
</table>

© Bright Computing, Inc.
12.6 Obtaining Monitoring Data Values

RequestDispatcher - - 0 0 0 -
RequestHandler - - 0 0 0 -

Storage | Elements | Disk size | Usage | Free disk
---------|----------|-----------|-------|----------
Mon::Storage::Engine | 0 | 0 B | 0.0% | -
Mon::Storage::Message | 0 | 0 B | 0.0% | -
Mon::Storage::RepositoryId | 0 | 0 B | 0.0% | -

12.6.7 Dropping Monitoring Data With The monitoringdrop Command

Monitoring data gathering can be restricted to certain nodes using node execution filtering and execution multiplexers. Entire data producers can also be disabled with the disable option in monitoring mode. However, restricting or disabling leaves historical samples in storage—the existing monitoring data values do not automatically get removed. So, in cmsh and Bright View the latest known monitoring data values then still show up, with a forever-increasing age.

If a data producer is removed, then the associated data values for its measurable or measurables are removed.

Alternatively, if adding execution filters to a monitoring data producer is intended to be a permanent change, then all previously collected data can be dropped for filtered nodes.

For example, if the ssh connectivity to only cloud nodes is to be checked:

Example

[root@bright82 ~]# cmsh
[bright82]% monitoring setup use ssh2node
[...->monitoring->setup[ssh2node]]% executionmultiplexers
[...->executionmultiplexers]% show
Type None
[...->executionmultiplexers]% use all nodes
[...->executionmultiplexers[All nodes]]% get types Node
[...->executionmultiplexers[All nodes]]% set types CloudNode
[...->executionmultiplexers*[All nodes*]]% commit

After this is set, the monitoring data values for a non-cloud node can be checked. The ssh2node health check data values are then seen to be getting older, without any more updates being added. These health check data values can then be dropped using the monitoringdrop command from within the device mode of cmsh command.

It is wise to run a dry-run operation first, in order to make sure that no data values are unintentionally removed:

Example

[root@bright82 ~]# cmsh
[bright82]% device use node001
[...->device[node001]]% latestmonitoringdata | grep ssh2node
ssh2node Network PASS 43m 38s filtered
[...->device[node001]]% monitoringdrop --dry-run --filtered

Entity Measurable
---------- -----------
node001 ssh2node

Removal of 1 entity, measurable pairs

[root@bright82 ~]# cmsh
[bright82]% device use node001
[...->device[node001]]% latestmonitoringdata | grep ssh2node

© Bright Computing, Inc.
The --force option can be used to remove non-filtered old data, such as data from a disabled measurable. This is also useful when correcting a bad metric script. After fixing the script, the old (incorrect) data can be dropped.

Example

[root@bright82 ~]# cmsh
[bright82]% device
[...->device]% monitoringdrop --category default my-metric --force
Removed 32 entity, measurable pairs

A reboot or CMDaemon restart is required for the node to start collecting data again on a non-filtered metric which has been dropped with the --force option.

12.6.8 Monitoring Suspension And Resumption—The monitoringsuspend And monitoringresume Commands

The monitoringsuspend command suspends monitoring. The monitoringresume command resumes monitoring.

When suspension is applied to a head node, the regular nodes simply continue sampling data up to a maximum of 1 million samples per node. The available backlog is fetched upon resumption.

Suspension can be used during benchmarking to measure the results of benchmarking runs without having monitoring get in the way.

Suspension can also be used as a quick sanity check during regular cluster operation, as a way for an administrator to see if it is monitoring that is consuming excessive resources, in comparison with the other processes on the system. For example, running it on a head node (some output omitted or elided):

Example

[root@head1 ~]# sar -b 1
Linux 2.6.32-754.10.1.el6.x86_64 (head1) 03/06/2019 _x86_64_ (28 CPU)

04:41:00 PM tps rtps wtps bread/s bwrtn/s
04:41:02 PM 1481.82 0.00 1481.82 0.00 24355.56
04:41:03 PM 849.49 0.00 849.49 0.00 10367.68
04:41:04 PM 509.00 0.00 509.00 0.00 4440.00
04:41:05 PM 709.90 0.00 709.90 0.00 5853.47
04:41:06 PM 1209.00 0.00 1209.00 0.00 18168.00
^C
[root@head1 ~]# cmsh
[head1]% device use master
[head1->device[head1]]% monitoringsuspend
suspend 14 on head1
[head1->device[head1]]% monitoringinfo

<table>
<thead>
<tr>
<th>Service</th>
<th>Queued</th>
<th>Handled</th>
<th>Cache miss</th>
<th>Stopped</th>
<th>Suspended</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mon::CacheGather</td>
<td>425</td>
<td>39,857</td>
<td>0</td>
<td>no</td>
<td>yes</td>
</tr>
<tr>
<td>Mon::DataConverter</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>no</td>
<td>yes</td>
</tr>
<tr>
<td>Mon::DataProcessor</td>
<td>0</td>
<td>6,609,369</td>
<td>0</td>
<td>no</td>
<td>yes</td>
</tr>
<tr>
<td>Mon::DataTranslator</td>
<td>0</td>
<td>311,658</td>
<td>0</td>
<td>no</td>
<td>yes</td>
</tr>
<tr>
<td>Mon::EntityMeasurableCache</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>no</td>
<td>yes</td>
</tr>
<tr>
<td>Mon::MeasurableBroker</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>no</td>
<td>yes</td>
</tr>
<tr>
<td>Mon::PerpetualTaskManager</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>no</td>
<td>yes</td>
</tr>
<tr>
<td>Mon::Replicate::Collector</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>yes</td>
<td>yes</td>
</tr>
<tr>
<td>Mon::Replicate::Combiner</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>yes</td>
<td>yes</td>
</tr>
</tbody>
</table>
In the preceding example monitoring is seen to be consuming significant resources.

After running `monitoringsuspend`, resuming monitoring should not be forgotten, and it should be done soon enough after suspension. If that is not done, then backlogged samples that exceed the limit of 1 million samples per node on the regular nodes would be lost. Resumption is carried out with:

**Example**

```bash
[root@head1 ~]# cmsh
[head1]% device use master
[head1->device[head1]]% monitoringresume
resume 14 on head1
```

**CMDaemon Directive Settings To Reduce Monitoring Resource Consumption**

The following CMDaemon directive changes may reduce the resource consumption due to monitoring:

- **Increasing the job account collection interval**: by increasing the value of the `JobsSamplingMetricsInterval` directive (page 654).

- **Disabling job information collection completely**: by setting the value of the `JobInformationDisabled` directive to 0 (page 659).

- **For the Slurm workload manager only, disabling job accounting**: by setting the value of the `SlurmDisableAccountingParsing` directive to 0 (page 654).

- **Reducing the duration for which job data is stored**: by reducing the value of the `JobInformationKeepDuration` directive (page 659).

**12.7 The User Portal**

**12.7.1 Accessing The User Portal**

The user portal is compatible with most browsers using reasonable settings. For Internet Explorer, version 9 or later is required.

The user portal is located by default on the head node. For a head node `bright82`, it is accessible to users for a login via a browser at the URL `http://bright82`, or more directly via `https://bright82:8081/userportal`. The state of the cluster can then be viewed by the users via an interactive interface.

The first time a browser is used to login to the portal, a warning about the site certificate being untrusted appears.

© Bright Computing, Inc.
The certificate is a self-signed certificate (the X509v3 certificate of Chapter 4 of the Installation Manual), generated and signed by Bright Computing, and the attributes of the cluster owner are part of the certificate. However, Bright Computing is not a recognized Certificate Authority (CA) like the CAs that are recognized by a browser, which is why the warning appears.

The SSL configuration file is located at /etc/httpd/conf.d/ssl.conf. Within ssl.conf, by default the PEM-encoded server certificate is set to /etc/pki/tls/certs/localhost.crt. The entry should be changed if generating and installing the certificate elsewhere. A key-based certificate can be used instead, with a private key then expected by the ssl.conf file at etc/pki/tls/private/localhost.key.

For a portal that is not accessible from the outside world, such as the internet, the warning about Bright Computing not being a recognized Certificate Authority is not an issue, and the user can simply accept the “untrusted” certificate.

For a portal that is accessible via the internet, some administrators may regard it as more secure to ask users to trust the self-signed certificate rather than external certificate authorities. Alternatively the administrator can replace the self-signed certificate with one obtained by a trusted recognized CA, for example the one at https://letsencrypt.org, if that is preferred.

12.7.2 Setting A Common Username/Password For The User Portal

By default, each user has their own username/password login to the portal. Removing the login is not possible, because the portal is provided by CMDaemon, and users must connect to CMDaemon.

A shared (common) username/password for all users can be set in the configuration file, common-credentials.json. The default username/password settings are blank, which means that common access is not enabled:

Example

```
{  
  "username": "",  
  "password": "" 
}
```

To enable common access:

- the common username and password must be added via cmsh or Bright View

Example

```
[root@bright82 ~]# cmsh  
[bright82]% user  
[bright82->user]% add forrestgump  
[bright82->user[forrestgump]]% set password  
enter new password:  
retype new password:  
[bright82->user*[forrestgump*]]% commit  
[bright82->user*[forrestgump*]]% quit
```

- the common username and password should be set in the appropriate place in the configuration file, common-credentials.json:

Example

```
[root@bright82 ~]# cat /cm/local/apps/cmd/etc/htdocs/userportal/assets/config/common-credentials.json  
{  
  "username": "forrestgump",  
  "password": "1forrest1" 
}
```
A minor stumbling block for the unwary administrator is:

If using Bright View, then if the password for the username has already been saved in the browser’s password manager before changing it in the configuration file, then the password saved in the browser’s password manager may need to be changed to the new one explicitly.

### 12.7.3 User Portal Home Page

#### User Portal Overview Page

The default user portal home page is the **Overview** page. This allows a quick glance to convey the most important cluster-related information for users (figure 12.24):

#### Overview

- **Message Of The Day**
  - This is the message of the day. Feel free to edit this to your liking (/assets/config/message-of-the-day.html).
- **Documentation**
  - Bright Computing website
  - Administrator manual
  - User manual
  - JSON API documentation
- **Cluster Overview**
  - **Uptime**: 0 days 0 hours 1 min
  - **Nodes**: 0 out of 0 total
  - **Devices**: 0 out of 0 total
  - **Cores**: 0 out of 2 total
  - **Users**: 0 out of 0 total
  - **CPU Load**: N/A
  - **Occupation Rate**: 0.25%
  - **Memory**: 0.00 B out of 0.00 B
  - **Swap**: 0.00 B out of 0.00 B total
  - **Load**: 1.00%
    - 0.00% user
    - 0.00% system
    - 0.00% nice
    - 100.00% other

Figure 12.24: User Portal: Overview Page

The following items are displayed on the overview page:

- **a Message Of The Day.** This can be edited in `/cm/local/apps/cmd/etc/htdocs/userportal/assets/config/message-of-the-day.html`
- links to the documentation for the cluster
- an overview of the cluster state, displaying some cluster parameters. By default, it is refreshed every 10s.

The user portal is designed to serve files only, and will not run executables such as PHP or similar CGI scripts. Monitoring and job accounting charts can be viewed on clicking upon the associated icons, and , at the top right corner of the user portal page.

#### User Portal Monitoring Page

The user portal’s Monitoring page allows charts of measurables to be plotted in a monitoring window in a very similar manner to how it is done in section 12.3.1.
12.8 Job Monitoring

Most HPC administrators set up device-centric monitoring to keep track of cluster node resource use. This means that metrics are selected for devices, and the results can then be seen over a period of time. The results can be viewed as a graph or data table, according to the viewing option chosen. This is covered in the other sections of this chapter.

The administrator can also select a job that is currently running, or that has recently run, and get metrics for nodes, memory, CPU, storage, and other resource use for the job. This is known as job monitoring, which is, as the term suggests, about job-centric rather than device-centric monitoring. It is covered in this section (section 12.8), and uses job metrics.

So far, monitoring as discussed in this chapter has been based on using devices or jobs as the buckets for which resource use values are gathered. Administrators can also gather, for resources consumed by jobs, the resources used by users (or any other aggregation entity) as the buckets for the values. This is typically useful for watching over the resources used by a user (or other aggregation entity) when jobs are run on the cluster. User-centric monitoring—or more generally, aggregation-centric monitoring—for jobs is termed job accounting and is covered in section 12.9.

12.8.1 Job Metrics Introduction

Job metrics collection uses cgroups (section 7.10). Thus each job should be associated by a workload manager with a unique cgroup automatically. Bright Cluster Manager configures the following workload managers to run jobs in cgroups:

- Slurm
- UGE
- LSF

Notes:
1. OGS for now does not support cgroups.
2. PBS Pro can be configured to run with cgroups, if the administrator enables cgroups by running the script: /cm/shared/apps/pbspro/var/cm/enable-cgroups.sh. For PBS Pro Community Edition the command is run automatically by wlm-setup. Hooks are provided for cgroups with the PBS Pro packages.

   Different workload managers use different cgroup controllers when creating a cgroup for a job. The controllers enable different metric collections to be carried out by the Linux kernel. In order to have the same set of collected metrics for each of the workload managers, Bright Cluster Manager configures systemd to mount the following cgroup controllers to the same directory:

   - blkio
   - cpuacct
   - memory
   - freezer

   For example: if Slurm jobs are configured to run on the regular nodes via wlm-setup, then the following cgroup directories structure becomes available for Slurm job metrics collection:

   © Bright Computing, Inc.
Example

```
[root@node001 ~]# cd /sys/fs/cgroup/
[root@node001 cgroup]# tree -d --charset=C -L 2 | grep blkio
|-- blkio    -> blkio,cpuacct,memory,freezer
|-- blkio,cpuacct,memory,freezer
|-- cpuacct   -> blkio,cpuacct,memory,freezer
|-- freezer   -> blkio,cpuacct,memory,freezer
|-- memory    -> blkio,cpuacct,memory,freezer
```

The directory

```
blkio,cpuacct,memory,freezer
```

is created, the appropriate cgroup controllers are mounted, the directory links are created, and this is all carried out automatically by systemd.

Example

```
[root@node001 ~]# grep JoinControllers /etc/systemd/system.conf
JoinControllers=blkio,cpuacct,memory,freezer
[root@node001 ~]#
```

The tables in Appendix G.1.5 list the job metrics that Bright Cluster Manager can monitor and visualize.

If job metrics are set up (section 12.8.3), then:

1. on virtual machines, block device metrics may be unavailable because of virtualization.

2. for now, the metrics are retrieved from cgroups created by the workload manager for each job. When the job is finished the cgroup is removed from the filesystem along with all the collected data. Retrieving the jobs metric data therefore means that CMDaemon must sample the cgroup metrics before the job is finished. If CMDaemon is not running during a time period for any reason, then the metrics for that time period cannot be collected, even if CMDaemon starts later.

3. block device metrics are collected for each block device by default. Thus, if there are \( N \) block devices, then there are \( N \) collected block device metrics. The monitored block devices can be excluded by configuration as indicated in section 12.8.3.

12.8.2 Job Information Retention

Each job adds a set of metric values to the monitoring data. The longer a job runs, the more data is added to the data. By default, old values are cleaned up from the database in order to limit its size. In Bright Cluster Manager 8.2 there are several advanced configuration directives to control the job data retention, with names and default values as follows:

<table>
<thead>
<tr>
<th>Advanced Configuration Directive</th>
<th>Default value</th>
</tr>
</thead>
<tbody>
<tr>
<td>JobInformationDisabled</td>
<td>0</td>
</tr>
<tr>
<td>JobInformationKeepDuration</td>
<td>2419200</td>
</tr>
<tr>
<td>JobInformationKeepCount</td>
<td>8192</td>
</tr>
<tr>
<td>JobInformationMinimalJobDuration</td>
<td>0</td>
</tr>
<tr>
<td>JobInformationFlushInterval</td>
<td>600</td>
</tr>
</tbody>
</table>

These directives are described in detail in Appendix C, page 659.
12.8.3 Job Metrics Sampling Configuration

Job metrics sampling can be configured to varying degrees. For clusters where hundreds of thousands of jobs are run in a day it often makes little sense to monitor jobs, and it is often helpful to disable the JobSampler and JobMetadataSampler data producers:

Example

[bright82]% monitoring setup
[bright82->monitoring->setup]% set jobsampler disabled yes
[bright82->monitoring->setup]% set jobmetadatasampler disabled yes
[bright82->monitoring->setup]% commit

If however CMDaemon is to keep the monitoring data, then the collection of job metrics is carried out from the cgroups in which a job runs. The administrator can tune some low level metric collection options for the JobSampler data producer in the jobmetricsettings submode:

Example

[bright82]% monitoring setup
[bright82->monitoring->setup]% use jobsampler
[bright82->monitoring->setup][JobSampler]% jobmetricsettings
[bright82->monitoring->setup][JobSampler]->jobmetricsettings]% show

Parameter Value

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Enable Advanced Metrics</td>
<td>Enables advanced metrics</td>
</tr>
<tr>
<td>Exclude Devices</td>
<td>Block devices for which job metrics will not collect metrics</td>
</tr>
<tr>
<td>Include Devices</td>
<td>If the list is not empty then only block device metrics for these devices will be collected, while for other devices the metrics will be skipped</td>
</tr>
<tr>
<td>Enable Advanced</td>
<td>Indicates whether advanced job metrics should be enabled (options: true, false)</td>
</tr>
</tbody>
</table>

...continues
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Exclude Metrics</td>
<td>List of metric names that should not be collected</td>
</tr>
<tr>
<td>Include Metrics</td>
<td>List of metric names that should be added to metric collection</td>
</tr>
</tbody>
</table>

The parameter Exclude Metrics can be used to exclude metrics that are currently enabled. For example, if advanced metrics collection is enabled then Exclude Metrics allows either default or advanced metrics to be excluded by name.

### 12.8.4 Job Monitoring In cmsh

Using cmsh, the following commands allow job metrics and historical information to be retrieved within jobs mode:

**filter**  The command provides filtered historic job-related information. It does not provide metrics data. The command and its options can be use to:

- retrieve running, pending, failed or finished jobs information
- select job data using regular expressions to filter by job ID
- list jobs by user name, user group, or workload manager

**Example**

```
[bright82->jobs(slurm)]% filter --limit 2
Job ID Job name User Queue Start time End time Nodes Exit code
------- --------- ---- ----- ------------------- ------------------- ------- ---------
10 slurm.job cmsupport defq 02/02/2016 13:58:22 03/02/2016 00:58:24 node001 0
11 slurm.job cmsupport defq 02/02/2016 13:58:22 03/02/2016 00:58:24 node001 0
```

The data shown is retrieved from the running workload manager, as well as from the accounting file or database maintained by the workload manager.

Further details of the options to filter can be seen by running `help filter`.

**dumpmonitoringdata**  The dumpmonitoringdata command for job metrics is somewhat similar to the dumpmonitoringdata command for device metrics (section 12.6.4), and shows monitoring data for a job ID over a period of time. Options allow metric values to be retrieved and filtered over a period of time. The user can also specify custom periods or select according to the user name of the job owner.

The usage of the dumpmonitoringdata command for job metrics is:

```
dumpmonitoringdata [OPTIONS] [<start-time> <end-time>] <metric> <job ID>
```

**Example**

```
[bright82->jobs(slurm)]% dumpmonitoringdata "16/02/02 19:30" "16/02/02 19:50" memory.usage 10
# From Tue Feb 2 19:30:00 2016 to Tue Feb 2 19:50:00 2016
# Nodes: node001
Time Value Info Message
-------------------------- ------------- ---------------
Tue Feb 2 19:30:00 2016 1.14062 MiB
Tue Feb 2 19:32:00 2016 1.14062 MiB
```

© Bright Computing, Inc.
The data is shown per node if the job uses several nodes.

Further details of the options to print monitoring data for job metrics can be seen by running `help dumpmonitoringdata` within `jobs` mode.

**statistics** The statistics command shows basic statistics for historical job information. It allows statistics to be filtered per user or user group, and workload manager. The format of the output can be specified with the `--format` option. The statistics can be grouped by hour, day, week or a custom interval.

**Example**

```
[bright82->jobs(slurm)]% statistics
```

```
Interval User Group Pending Running Finished Error Nodes
--------- ------- ------- -------- -------- --------- ------ ----- 
N/A 0 7 32 0 34
```

Further details of the options to `statistics` can be seen by running `help statistics`.

### 12.9 Job Accounting

In addition to the concept of device-based monitoring (described in most of this chapter), or the concept of job-based metrics (section 12.8 of this chapter), there is also the concept of aggregation-based metrics for resources used during jobs. This last one is typically user-based metrics for resources used during jobs. Aggregation-based metrics for jobs is more conveniently called job accounting, and its concept and implementation are described in this section (section 12.9).

Job accounting is thus about the accounting of resources used by jobs. Typically, for jobs, resource usage can be presented per user. For example, if there are jobs in a queue that are being processed, the jobs can be listed:

**Example**

```
[bright82->jobs(slurm)]% list | head
```

```
Type Job ID User Queue Running time Status Nodes
------------ ------------ ------ ------- ------------ ---------- -------------------
Slurm 1325 tim defq 1m 2s COMPLETED node001..node003
Slurm 1326 tim defq 1m 1s COMPLETED node001..node003
Slurm 1327 tim defq 1m 2s COMPLETED node001..node003
Slurm 1328 tim defq 32s RUNNING node001..node003
Slurm 1329 tim defq 0s PENDING
```

The resource usage statistics gathered per user, for example a user `tim`, can then be analyzed and visualized using the job accounting interface of Bright View (section 12.9.3).
12.9.1 Labeled Entities

In job accounting, job metrics during a run are tagged with extra labels, such as the job ID, host name, and the user running the job. The modified job metrics object that is tagged in this way then becomes a job accounting-related object, called a labeled entity.

Administrators interested in using job accounting can simply skip ahead and start reading about the Bright View job accounting interface in 12.9.3, and just explore it directly. Those who would prefer some background on how job accounting is integrated with Bright Cluster Manager and PromQL, can continue reading this section (12.9.1) and the next one (12.9.2).

Dataproducers For Labeled Entities

To view labeled entities in cmsh, the path to the labeledentity submode is:

```
cmsh → monitoring → labeledentity
```

The default, existing labeled entities are created from the built-in JobSampler and JobMetadataSampler dataproducers when a job is run. Custom samplers, of type prometheus, can be used to create further custom labeled entities. A custom sampler dataproducer, for example customsamplerextras, can be created from the monitoring setup mode of cmsh as follows:

Example

```
[bright82] monitoring setup
[bright82->monitoring->setup]% add prometheus customsamplerextras
[bright82->monitoring->setup*[customsamplerextras*]]% ...
```

The customsamplerextras dataproducer can now have its properties configured and committed as described in section 12.5.4.

PromQL And Labeled Entities

Labeled entities can be used by administrators to help create and debug job-related queries in the Prometheus query language, PromQL. PromQL is a part of the Prometheus monitoring and alerting toolkit (https://prometheus.io). Basic PromQL documentation is available at https://prometheus.io/docs/prometheus/latest/querying/basics/.

Job IDs And Labeled Entities

Each job ID has a number of labeled entities associated with it. Since the number of labeled entities scales with the number of nodes and jobs, the number of labeled entities can be very large. Therefore, if examining these entities using the CMDaemon front ends such as cmsh or Bright View, then filtering or sorting the output is useful. For example, labeled entities associated with node001, and with the JobSampler data producer, and with job 1329 from the preceding output, could be viewed by filtering the full list of labeled entities as follows (output truncated and ellipsized):

Example

```
[bright82->monitoring->labeledentity]% list|head -2; list|grep 'job_id="1329"' |grep node001 |grep JobSampler
```

```
Index Name (key) ...
------ --------------------------------------------------------------------------------------...
45446 hostname="node001",job="JobSampler",job_id="1329",wlm="slurm" ...
45447 device="vda",hostname="node001",job="JobSampler",job_id="1329",wlm="slurm" ...
45448 device="vda",hostname="node001",job="JobSampler",job_id="1329",mode="read",wlm="slurm"... ...
```

Measurables And Labeled Entities

The measurables (metrics) for an entity can be listed with the measurables (metrics) command. For a particular entity with a JobSampler property and index value of 45447, the command can be run as follows:
In the labeledentity mode of cmsh, the measurables listing command, which lists the measurables for labeled entities, should not be confused with the measurable navigation command, which brings the administrator to the measurable submode under the main monitoring mode.

12.9.2 PromQL Queries

The Default PromQL Queries...

By default there are several predefined PromQL queries already available. The queries can be listed from the query submode:

Example

The queries can be conceptually divided into their classes, which at the time of writing (August 2019) are: accounting, jobs, ceph, and cluster.

Metrics of all classes except for accounting are sampled every 15 minutes by default.

A metric in the accounting class query is evaluated (interpolated) from existing values. These existing values are raw samples gathered from 1 week ago, up to the time when the query is evaluated. The _1w ending to the name indicates the 1 week period.
...And A Short Description Of Them

The description of each query can be listed with a little cmsh and unix text utility juggling:

Example

```bash
[bright82->monitoring->query]% foreach * (get name; get description) | paste - - | expand -t 37
users_job_wall_clock_seconds_1w  Wall clock time used by users for the past 7 days
users_job_wasted_cpu_seconds_1w  CPU seconds allocated but not used by users for the past 7 days
users_job_effective_cpu_seconds_1w  CPU seconds effectively used by users for the past 7 days
users_job_power_usage_watts_1w  Power usage by users during the last week in Joule (Watt seconds)
users_job_memory_usage_bytes_1w  Total memory usage by users during the last week in Byte seconds
users_job_io_bytes_1w  Total I/O by users during the last week in Bytes
account_job_effective_cpu_seconds_1w  CPU seconds effectively used by account for the past 7 days
account_job_memory_usage_bytes_1w  Total memory usage by account during the last week in Bytes
account_job_power_usage_watts_1w  Power usage by account during the last week in Joule (Watt seconds)
account_job_waiting_seconds_1w  Total waiting time for account jobs in seconds during the past 7 days
account_job_wall_clock_seconds_1w  Wall clock time used by account for the past 7 days
account_job_wasted_cpu_seconds_1w  CPU seconds allocated but not used by account for the past 7 days
users_job_allocated_nodes  Number of nodes allocated by users
users_job_cpu_usage  Effective CPU usage by users
users_job_cpu_usage_watts  Instantaneous power usage by users in Watts. Calculated by assigning
users_job_waiting  Number of jobs currently waiting for every user
users_job_memory_bytes  Current memory consumption for user jobs in Bytes
users_job_io_bytes_per_second  Current I/O for user jobs in B/s
cluster_cpu_usage_percent  CPU usage percentage over all nodes up
storage_ceph_osd_bytes_used  The amount of disk space used across all OSDs
storage_ceph_host_apply_latency  The average apply latency on each host running Ceph OSDs
storage_ceph_host_commit_latency  The average commit latency on each host running Ceph OSDs
storage_ceph_host_cpu_usage  CPU usage on each host running Ceph OSDs
storage_ceph_osd_op_ratio  The number of operations per second on each Ceph OSD
```

The listings give an idea of what the query does.

For example, for the `users_job_cpu_usage` utility, the idea is that it shows the CPU usage for jobs for each user.

Modifying The Default PromQL Query Properties

The properties of a particular query can be shown and modified:

Example

```bash
[bright82->monitoring->query]% use users_job_cpu_usage
[bright82->monitoring->query[users_job_cpu_usage]]% show
Parameter       Value
Access          Public
Class           jobs
Description     Effective CPU usage by users
End time        now
Interval        15m
Name            users_job_cpu_usage
Notes           <0 bytes>
PromQL Query    <131 bytes>
```
The PromQL query code itself is typically a few lines long, and can also be viewed and modified using `get` and `set`.

**An Example PromQL Query, Properties, And Disassembly**
The `users_job_cpu_usage` query is a standard predefined query, and is used as an example here. The query shows the CPU usage by a user around the time the sample was taken. It is sometimes called an “instantaneous” value. However, it is not that instantaneous, because its value is calculated by taking samples of the CPU usage over the last 10 minutes of the job run rather than at the query time. The code for the query can be viewed with:

**Example**

```bash
[bright82->monitoring->query] % get users_job_cpu_usage promqlquery
sum by(user) (irate(job_cpuacct_usage_seconds[10m])
  * on(job_id, hostname) group_right()
  (job_metadata_is_running)
)
```

For those unfamiliar with PromQL, some disassembly of the `users_job_cpu_usage` query is helpful. Terminology used by PromQL and Bright Cluster Manager, for the pieces used to build the query, is listed in the following table:

<table>
<thead>
<tr>
<th>PromQL Terminology</th>
<th>Example</th>
<th>Bright Cluster Manager Terminology</th>
</tr>
</thead>
<tbody>
<tr>
<td>Query</td>
<td>users_job_cpu_usage</td>
<td>PromQL query</td>
</tr>
<tr>
<td>Instant query</td>
<td>job_cpuacct_usage_seconds</td>
<td>Metric (from JobSampler dataproducer, belonging to the Prometheus class)</td>
</tr>
<tr>
<td>Range vector</td>
<td>job_cpuacct_usage_seconds[10m]</td>
<td>Metric samples over a time span</td>
</tr>
</tbody>
</table>

As was mentioned before: job account metrics, unlike traditional metrics, are not directly associated with the device-related objects. For such metrics, the monitoring data command `dumpmonitoringdata` is therefore not accessed in `cmsh` from `device` mode or `category` mode.

Instead, the Prometheus metric `job_cpuacct_usage_seconds`, for example, is accessed via the `labeledentities` mode.

The properties and interpolated values at a particular instant of time for the metric can be accessed via an `instantquery` such as (some output excised for clarity):

**Example**

```bash
[bright82->monitoring->labeledentity] % instantquery job_cpuacct_usage_seconds
__name__ hostname job job_id uid user wlm Timestamp Value
---------------------------------------- ----------- ------- ---- ----- ----- --------- ----- 
job_cpuacct_usage_seconds node001 JobSampler 623 1002 tony slurm 15:52:15 129
job_cpuacct_usage_seconds node002 JobSampler 624 1002 tony slurm 15:52:15 71
...
```

The properties and interpolated values of the metric over a range of time can be accessed via a `rangequery` such as (some output excised for clarity):

**Example**
12.9 Job Accounting

The association of the instantquery and rangequery output with job accounts is because its data producer is JobSampler.

Further options for the instantquery and rangequery commands can be found in their help texts within cmsh.

Aside: Getting Raw Values For A Prometheus Class Metric

The PromQL language is aimed at providing an overall view of jobs and resource usage. The actual individual raw values that Prometheus metrics are built on—the entries in the Time Series Database (TSDB)—are not regarded as being important for the end user. The emphasis in PromQL is on seeing the values as seen by statistical reworking.

This section, which is about the raw TSDB values, is thus provided as background information for administrators who would anyway like to see what the raw values look like.

Raw values of the metric for a job ID can be accessed by using the index of the labeled identity that is associated with that job ID. For example, job ID 624 can have its index found with some grepping (some output elided or excised for clarity):

```
Example
[bright82->monitoring->labeledentity]% list|head -2; list|grep ' hostname='| grep 'job_id="624"'
Index Name (key) Introduction Last used
----- ----------------------------------------------------- ------------ ---------
4060 hostname="node001",job="JobSampler",job_id="624", ... 13:50:03 16:00:03

[bright82->monitoring->labeledentity]%
```

The index for the job ID 624 is 4060. The job ID can be used by the dumpmonitoringdata command to show the series raw values along with their time stamps:

```
Example
[bright82->monitoring->labeledentity]% dumpmonitoringdata -24h now job_cpuacct_usage_seconds 4060
```

© Bright Computing, Inc.
These raw values are the values that are used for interpolation during PromQL queries. The label names for job samples can be seen using the index:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Index</td>
<td>4060</td>
</tr>
<tr>
<td>Introduction</td>
<td>Thu, 01 Aug 2019 16:08:03 CEST</td>
</tr>
<tr>
<td>Last used</td>
<td>Thu, 01 Aug 2019 16:30:03 CEST</td>
</tr>
<tr>
<td>Name</td>
<td>hostname=&quot;node001&quot;, job=&quot;JobSampler&quot;, job_id=&quot;624&quot;, uid=&quot;1002&quot;, user=&quot;tony&quot;, wlm=&quot;slurm&quot;</td>
</tr>
<tr>
<td>Permanent</td>
<td>no</td>
</tr>
<tr>
<td>Revision</td>
<td></td>
</tr>
</tbody>
</table>

*An Example PromQL Query, Properties, And Disassembly (Continued)*

Getting back from the aside about raw values, and continuing on with the example PromQL query from the start of this section (page 534), the query code for the query `users_job_cpu_usage` was:

```promql
sum by(user) (irate(job_cpuacct_usage_seconds[10m]) on(job_id, hostname) group_right())
```

With the necessary background explanations having been carried out, the disassembly of this query can now be done:

The core of the query is built around the job sampler metric `job_cpuacct_usage_seconds`. The `irate` measurement in this case calculates the rate of change based on the last two most recent values in the Prometheus range vector. The Prometheus range vector is formed from the Prometheus instant query by using the square brackets with a time value enclosed (`job_cpuacct_usage_seconds[10m]`). The Prometheus instant query is, as the terminology table earlier pointed out, the Prometheus version of the job sampler metric.

Getting back to the range vector, a range vector in general is a series of values formed from the corresponding Prometheus instant query. With the instant query being `job_cpuacct_usage_seconds` here, the range vector is formed over a span of 10 minutes.

After the `irate` function has taken the average, the resultant is what in PromQL is called an instant vector value. This consists of data in the form {CPU seconds consumed during period, timestamp associated with time period sample}. The instant vector value is then joined against each vector element of the pair `job_id`, `hostname` to generate the labeled identifier for the job running on the node. The `group_right` of the result uses the `job_id` and `hostname` as the leading labels in the label identifiers. The `job_metadata_is_running` function means that values are generated only while `job_metadata` is running. The `sum by(user)` function means that sum of the result is aggregated per user.

Visualisation based on the result is most easily carried out by plotting job CPU usage for each user against time in the period specified, which can be done in a more user-friendly way with Bright View.

The `users_job_wall_clock_seconds_1w` query, is similar, and can be used to plot wall clock seconds consumed by a user over the last week:

```promql
sum by(user) (max_over_time(job_metadata_running_seconds[1w]) on(job_id) group_right())
```

© Bright Computing, Inc.
Predefined queries can be executed in the labeledentity mode with the -q option:

```
[bright82->monitoring->labeledentity]% instantquery -q users_job_wall_clock_seconds_1w
```

<table>
<thead>
<tr>
<th>user</th>
<th>Timestamp</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>alice</td>
<td>Mon Jun 24 10:19:34 2019</td>
<td>28.787</td>
</tr>
<tr>
<td>charline</td>
<td>Mon Jun 24 10:19:34 2019</td>
<td>102.83</td>
</tr>
<tr>
<td>eve</td>
<td>Mon Jun 24 10:19:34 2019</td>
<td>58.574</td>
</tr>
<tr>
<td>frank</td>
<td>Mon Jun 24 10:19:34 2019</td>
<td>59.575</td>
</tr>
</tbody>
</table>

If Bright View is used instead of cmsh, then Prometheus queries can be accessed via the clickpath:

Monitoring → PromQL Queries

### 12.9.3 Job Accounting In Bright View

Job accounting can be viewed within Bright View’s accounting mode by clicking on the calculator icon at the top right hand corner of the Bright View standard display (figure 12.5). An accounting panel is then displayed (figure 12.25), within a dashboard:

![Figure 12.25: Job Accounting: Panel](image)

**Management And Use Of The Accounting Panel**

A new panel can be added by clicking on the button in the dashboard panel.

By clicking on the gear icon in the dashboard panel, the accounting panel layout can be changed to allow other arrangements of the panels, or to allow more panels, with other queries.

An accounting panel itself allows a previously-created query to be selected and executed. A PromQL query can be executed with default parameters by just clicking on the Run button.

The box to the right of the Run button shows the query name, and its description.

How to run and view PromQL queries in the Bright View accounting panel is described in this section (12.9.3). The query specification in Bright View is described in more detail in section 12.9.4.

© Bright Computing, Inc.
PromQL queries input in the accounting panel: Predefined PromQL queries (page 532) can be viewed via the dropdown list that appears under Saved queries. By default, the panel has the query `account_job_effective_cpu_seconds_1w` selected.

- In the default Basic mode of the panel, multiple queries of the same class can be ticked per panel (figure 12.26). To tick a new class of query in the same panel, all the queries that do not match the new class must first be unticked. In the panel of figure 12.26 the class is `accounting`.

PromQL queries run results display in the accounting panel: Run results can be displayed using the row toggle or the plot toggle.

- The row toggle toggles displaying the result as rows of data per user, cluster, account, or other aggregate (figure 12.25)

- The plot toggle toggles displaying the result as a pie chart, or as a standard x-y plot:
  - The pie chart, or doughnut chart, is available for a PromQL Instant mode query (figure 12.27). The chart shows how much of the resource is used at that instant per user, or per cluster, or per account, or per other aggregate.
How much of the resource is used can be displayed upon the chart in figures, either as the amount itself, or as a percentage of the total aggregate amount.

The chart can display aggregates using a maximum of 10 slices, by default. If needed, the right amount of the smallest extra slices are grouped together as one slice, others, so that the maximum number is not exceeded.

Figure 12.27: Job Accounting: Doughnut Display At An Instant

- The x-y plot is available for a PromQL Range mode query (figure 12.28).
  The plot shows resource use versus time. The plot can be per user, cluster, account, or other aggregate.
  A particular user or other aggregate can be highlighted on the plot by placing the mouse pointer over the appropriate legend color icon. The time scale can be zoomed in or out with the mouse scrollbar.
How Bright View is used to take PromQL query inputs, and present PromQL query results, has been covered in this section without going into much detail on PromQL specification. PromQL specification is covered in more detail in the next section.

12.9.4 PromQL Query Modes And Specification In Bright View

Clicking on the gear icon for the Accounting Panel (figure 12.25) allows the Accounting widget mode to be toggled to Advanced. Options that can then be modified include:

- the Query Type: This can be either Instant or Range.

An Instant query type: is executed at a particular instant. It outputs a single number for each entity (usually a user). The number is the interpolated data value for the PromQL query at that time. Each data value obtained by the PromQL query is typically an aggregate of the data values over a preceding time period, where the preceding time period is defined in the PromQL query. The single number obtained by the query for each entity is typically used to summarize a value at that instant for that entity.

An example of the Instant query type is `users_job_waiting_seconds_1w`. This provides the total waiting time, in seconds, for each user, for the jobs the users ran during the past week. The waiting time is calculated at the instant specified for the query.

If the Time parameter for the query is kept as the default value of now, then the query provides the latest value for each user.

If the Time parameter is set to a time in the past, then the query picks up the value at that time in the past, or provides an interpolated value for that time in the past, for each user.

The display can be presented in rows of user, value pairs. As a plot, the display can be presented as a pie chart, which each slice being proportional to the value for the particular user compared
12.9 Job Accounting

with the total value for all the users. Instead of a pie chart, the visualization can instead be a ring (or doughnut) chart, which is really just a pie chart with a hole in the centre.

PromQL allows a variety of time specifications. The interface validates whatever the interface user types in, and there is a calendar widget that allows an absolute time to be specified. Some useful time specifications are:

Example

<table>
<thead>
<tr>
<th>Time specification</th>
<th>What time is meant</th>
</tr>
</thead>
<tbody>
<tr>
<td>now</td>
<td>at the time it is run</td>
</tr>
<tr>
<td>now-30m</td>
<td>30 minutes ago</td>
</tr>
<tr>
<td>now-1h</td>
<td>one hour ago</td>
</tr>
<tr>
<td>now-1h/h</td>
<td>an hour ago, starting at the start of that hour</td>
</tr>
<tr>
<td>now-2d</td>
<td>48 hours ago</td>
</tr>
<tr>
<td>now-0d/d</td>
<td>today’s midnight (the most recently-passed midnight)*</td>
</tr>
<tr>
<td>now-1d/d</td>
<td>yesterday’s midnight**</td>
</tr>
<tr>
<td>now-2d/d</td>
<td>day before yesterday’s midnight</td>
</tr>
<tr>
<td>Thursday, July 11, 2019 17:00:00</td>
<td>an absolute time (as set by the calendar widget)</td>
</tr>
</tbody>
</table>

*The meaning can be understood by looking at how /d operates. It rounds off the day by truncation, which in the case of now-0d/d means the midnight prior to now.

**Similarly, yesterday’s midnight is the midnight immediately prior to 24 hours ago.

The time units for PromQL are (https://prometheus.io/docs/prometheus/latest/querying/basics/#range-vector-selectors):

- s - seconds
- m - minutes
- h - hour
- d - day
- w - week
- M - month (31 days). M is not a PromQL unit, but can be used in Bright Cluster Manager as a handy alias for a time period of 31 days. It is invariant, and does not vary in length with the month.
- y - year

The - operator in Prometheus is an offset operator, used only after now in the time specification field. The /<time unit> syntax implies a start at that unit of time.

Thus if, for example, now-1d/d is set as the end time, then when the query runs, it picks up the values at “yesterday’s midnight.” For each user, the value is the number of seconds the job of the user was in a wait state for the week prior to yesterday’s midnight. It rounds off the day by truncation, which in the case of now-1d/d means the midnight prior to 24 hours ago. Varying the user means that the number of seconds varies accordingly.

The results of a run can be

- displayed in rows as values in Bright View itself (figure 12.29)
- downloaded as a CSV file
- downloaded as an Excel spreadsheet.

© Bright Computing, Inc.
A Range query type: This query is executed over a time period. It fetches a value for each interval within that time range, and then plots these values.

An example of the Range query type is users_job_memory_bytes. This fetches the instantaneous memory usage for jobs over intervals, grouped per user during that interval (figure 12.29). The query submitter can set the Start Time and End Time parameters. However, very long ranges can be computationally expensive, and the graph resolution may not be ideal when zoomed in.

![Figure 12.29: Job Accounting: User Values Over A Range](image)

- the PromQL Query: Within advanced mode, the PromQL query itself can be edited.
- the Interval: When editing a Range query type via the Saved query option, the interval for a range query run can be set. Lower numbers lead to smaller intervals, hence higher resolution and higher execution times. A value of 0 means that a reasonable default interval is attempted. Choosing a interval that is less than the sampling time of the metric is a bit pointless, and tends to lead to data values that display non-smooth behaviour.

12.10 Event Viewer

Monitoring in Bright Cluster Manager is normally taken by developers to mean how sampling with data producers is handled. However, cluster administrators using this manual typically consider watching and handling events in Bright Cluster Manager to also be a part of a more general concept of monitoring. This manual is aimed at cluster administrators, and therefore this section on event viewing and handling is placed in the current Cluster Monitoring chapter.

Bright Cluster Manager events can be handled and viewed in several ways.

12.10.1 Viewing Events In Bright View

In Bright View, events can be viewed by clicking on the Events icon of figure 12.5. This opens up a window with a sortable set of columns listing the events in the events log, and with by default with the most recent events showing up first.

12.10.2 Viewing Events In cmsh

The events command is a global cmsh command. It allows events to be viewed at several severity levels (section 12.2.7), and allows old events to be displayed. The usage and synopsis of the events command is:
Usage: 
```
events
events on [broadcast|private]
events off [broadcast|private]
events level <level>
events clear
events details <id> [<id>]
events <number> [level]
events follow
```

Arguments:

```
level info,notice,warning,error,alert
```

Running the command without any option shows event settings, and displays any event messages that have not been displayed yet in the session:

**Example**

```
[bright82->device]# events
Private events:  off
Broadcast events: on
Level:          notice
custom .......................[  RESET  ] node001
```

Running the command with options allows the viewing and setting of events as follows:

- **on** [broadcast|private]: event messages are displayed as they happen in a session, with `cmsh` prompts showing in between messages:
  - If only `on` is set, then all event messages are displayed as they happen:
    - either to all open `cmsh` sessions, and also in Bright View event viewer panes, if the event or its trigger has the “broadcast” property.
    - or only in the `cmsh` session that is running the command, if the event or its trigger has the “private” property.
  - If the further option `broadcast` is set, then the event message is displayed as it happens in all open `cmsh` sessions, and also in all Bright View event viewer panes, if the event or its trigger has the “broadcast” property.
  - If the further option `private` is set, then the event message is displayed as it happens only in the `cmsh` session that ran the command, if the event or its trigger has the “private” property.

- **off** [broadcast|private]: disallows viewing of event messages as they happen in a session. Event messages that have not been displayed due to being forbidden with these options, are displayed when the `events` command is run without any options in the same session.
  - If only `off` is set, then no event message is displayed as it happens in the session. This is regardless of the “broadcast” or “private” property of the event or its trigger.
  - If the further option `broadcast` is set, then the event message is not displayed as it happens, if the event or its trigger has the “broadcast” property.
  - If the further option `private` is set, then the event message is not displayed as it happens, if the event or its trigger has the “private” property.

- **level** <info|notice|warning|error|alert>: sets a level. Messages are then displayed for this and higher levels.
- **clear**: clears the local `cmsh` event message cache. The cache indexes some of the events.
• **details <id>:** shows details for a specific event with the index value of <id>, which is a number that refers to an event.

• **<number> [info|notice|warning|error|alert]:** shows a specified <number> of past lines of events. If an optional level (info, notice,...) is also specified, then only that level and higher (more urgent) levels are displayed.

• **follow:** follows event messages in a cmsh session, similar to `tail -f /var/log/messages`. This is useful, for example, in tracking a series of events in a session without having the cmsh prompt showing. The output can also be filtered with the standard unix text utilities, for example: `events follow | grep node001`

A common example of events that send private messages as they happen are events triggered by the `updateprovisioners` command, which has the “private” property. The following example illustrates how setting the event viewing option to private controls what is sent to the cmsh session. Some of the output has been elided or truncated for clarity:

**Example**

```bash
[bright82->softwareimage]% events on private
Private events: on
[bright82->softwareimage]% updateprovisioners
Provisioning nodes will be updated in the background.
[bright82->softwareimage]% Tue Apr 29 01:19:12 2014 [notice] bright82: Provisioning started: sendi...
[bright82->softwareimage]% Tue Apr 29 01:19:52 2014 [notice] bright82: Provisioning completed: sendi...
[bright82->softwareimage]% !#events were indeed seen in cmsh session
[bright82->softwareimage]% !#now block the events and rerun update:
[bright82->softwareimage]% events off private
Private events: off
[bright82->softwareimage]% updateprovisioners
Provisioning nodes will be updated in the background.
[bright82->softwareimage]% !#let this 2nd update run for a while
[bright82->softwareimage]% !#(time passes)
[bright82->softwareimage]% !#nothing seen in cmsh session.
[bright82->softwareimage]% !#show a 2nd update did happen:
[bright82->softwareimage]% Tue Apr 29 01:19:12 2014 [notice] bright82: Provisioning started: sendi...
[bright82->softwareimage]% Tue Apr 29 01:19:52 2014 [notice] bright82: Provisioning completed: sendi...
[bright82->softwareimage]% Tue Apr 29 01:25:37 2014 [notice] bright82: Provisioning started: sendi...
[bright82->softwareimage]% Tue Apr 29 01:26:01 2014 [notice] bright82: Provisioning completed: sendi...
```

**12.10.3 Using The Event Bucket From The Shell For Events And For Tagging Device States**

**Event Bucket Default Behavior**

The Bright Cluster Manager `event bucket` accepts input piped to it, somewhat like the traditional unix “bit bucket”, `/dev/null`. However, while the bit bucket simply accepts any input and discards it, the event bucket accepts a line of text and makes an event of it. Since the event bucket is essentially an event processing tool, the volumes that are processed by it are obviously less than that which `/dev/null` can handle.

By default, the location of the event bucket is at `/var/spool/cmd/eventbucket`, and a message can be written to the event pane like this:

**Example**

© Bright Computing, Inc.
This adds an event with, by default, the info severity level, to the event pane, with the **InfoMessage** “Some text”.

### 12.10.4 InfoMessages

InfoMessages are optional messages that inform the administrator of the reason for the status change of a measurable, or an event in the cluster.

Measureable scripts can use file descriptor 3 within their scripts to write an InfoMessage:

**Example**

```
echo "Drive speed unknown: Reverse polarity" >&3
```

### Event Bucket Severity Levels

To write events at specific severity levels (section 12.2.7), and not just at the info level, the appropriate text can be prepended from the following to the text that is to be displayed:

- `EVENT_SEVERITY_DEBUG`
- `EVENT_SEVERITY_INFO`
- `EVENT_SEVERITY_NOTICE`
- `EVENT_SEVERITY_WARNING`
- `EVENT_SEVERITY_ERROR`
- `EVENT_SEVERITY_ALERT`

**Example**

```
echo "EVENT_SEVERITY_ERROR: An error line" > /var/spool/cmd/eventbucket
```

The preceding example displays an output notification in the Bright View event viewer as shown in figure 12.30:

![Figure 12.30: Bright View Monitoring: Event Bucket Message Example](image)

### Event Bucket Filter

Regex expressions can be used to conveniently filter out the user-defined messages that are about to go into the event bucket from the shell. The filters used are placed in the event bucket filter, located by default at `/cm/local/apps/cmd/etc/eventbucket.filter`.

### Event Bucket CMDaemon Directives

The name and location of the event bucket file and the event bucket filter file can be set using the `EventBucket` and `EventBucketFilter` directives from the CMDaemon configuration file directives (Appendix C).

### Adding A User-Defined Message To A Device State With The Event Bucket

While the event bucket is normally used to send a message to the event viewer, it can instead be used to add a message to the state of a device. The line passed to the `echo` command then has the message and device specified in the following format:

```
STATE.USERMESSAGE[.device]:[message].
```

The device can be anything with a status property, such as, for example, a node, a switch, or a chassis.
Example

echo "STATE.USERMESSAGE.node001: just right" > /var/spool/cmd/eventbucket

The state then shows as:

cmsh -c "device ; status node001"
node001 .................. (just right) [ UP ]

If the device is not specified, then the current host of the shell that is executing the echo command is used. For example, running these commands from the head node, bright82, as follows:

Example

echo "STATE.USERMESSAGE: too hot" > /var/spool/cmd/eventbucket
ssh node001 'echo "STATE.USERMESSAGE: too cold" > /var/spool/cmd/eventbucket'

yields these states:

cmsh -c "device ; status bright82"
bright82 .................. (too hot) [ UP ]
cmsh -c "device ; status node001"
node001 .................. (too cold) [ UP ]

The added text can be cleared with echoing a blank message to that device. For example, for node001 that could be:

echo "STATE.USERMESSAGE.node001:" > /var/spool/cmd/eventbucket

© Bright Computing, Inc.
Day-to-day Administration

This chapter discusses several tasks that may come up in day-to-day administration of a cluster running Bright Cluster Manager.

13.1 Parallel Shells: pdsh And pexec

What pdsh And pexec Do

The cluster management tools include two parallel shell execution commands:

- pdsh (parallel distributed shell, section 13.1.1), runs from within the OS shell. That is, pdsh executes its commands from within bash by default.
- pexec (parallel execute, section 13.1.2), runs from within CMDaemon. That is, pexec executes its commands from within the cmsh front end.

A one-time execution of pdsh or pexec can run one or more shell commands on a group of nodes in parallel.

A Warning About Power Surge Risks With pdsh And pexec

Some care is needed when running pdsh or pexec with commands that affect power consumption. For example, running commands that power-cycle devices in parallel over a large number of nodes can be risky because it can put unacceptable surge demands on the power supplies.

Within cmsh, executing a power reset command from device mode to power cycle a large group of nodes is much safer than running a parallel command to do a reset using pdsh or pexec. This is because the CMDaemon power reset powers up nodes after a deliberate delay between nodes (section 4.2).

Which Command To Use Out Of pdsh And pexec

The choice of using pdsh or pexec commands is mostly up to the administrator. The only time that running pdsh from bash is currently required instead of running pexec from within cmsh, is when stopping and restarting CMDaemon on a large number of regular nodes (section 2.6.1). This is because a command to stop CMDaemon on a regular node, that itself relies on being run from a running CMDaemon on a regular node, can obviously give unexpected results.

13.1.1 pdsh In The OS Shell

Packages Required For pdsh

By default, the following packages must be installed from the Bright Cluster Manager repositories to the head node for pdsh to function fully:

- pdsh
- genders

© Bright Computing, Inc.
The `pdsh` utility is modular, that is, it gets extra functionality through the addition of modules. The `genders` package is used to generate a default `/etc/genders` configuration file. The file is used to decide what and how nodes are to be used or excluded by default, when used with `pdsh`. Configuration details can be found in `man pdsh(1)`. The configuration can probably best be understood by viewing the file itself and noting that Bright Cluster Manager in the default configuration associates the following genders with a list of nodes:

- **all**: all the nodes in the cluster, head and regular nodes.
- **category=default**: the nodes that are in the default category
- **computenode**: regular nodes
- **headnode**: node or nodes that are head nodes.

In a newly-installed cluster using default settings, the genders `category=default` and `computenode` have the same list of nodes to begin with.

The default `/etc/genders` file has a section that is generated and maintained by CMDaemon, but the file can be altered by the administrator outside the CMDaemon-maintained section. However, it is not recommended to change the file manually frequently. For example, tracking node states with this file is not recommended. Instead, the package `pdsh-mod-cmupdown` provides the `-v` option for node state tracking functionality, and how to use this and other `pdsh` options is described in the next section.

### `pdsh` Options

In the OS shell, running `pdsh -h` displays the following help text:

```
Usage: pdsh [-options] command ...
-S              return largest of remote command return values
-h              output usage menu and quit
-V              output version information and quit
-q              list the option settings and quit
-b              disable ^C status feature (batch mode)
-d              enable extra debug information from ^C status
-l user         execute remote commands as user
-t seconds     set connect timeout (default is 10 sec)
-u seconds     set command timeout (no default)
-f n           use fanout of n nodes
-w host,host,... set target node list on command line
-x host,host,... set node exclusion list on command line
-R name        set rcmd module to name
-M name,...    select one or more misc modules to initialize first
-N              disable hostname: labels on output lines
-L              list info on all loaded modules and exit
-v              exclude targets if they are down
-g query,...   target nodes using genders query
-X query,...   exclude nodes using genders query
-F file        use alternate genders file 'file'
-i             request alternate or canonical hostnames if applicable
-a             target all nodes except those with "pdsh_all_skip" attribute
-A             target all nodes listed in genders database
available rcmd modules: ssh,exec (default: ssh)
```
Further options and details are given in man pdsh(1).

Examples Of pdsh Use
For the examples in this section, a cluster can be considered that is set up with two nodes, with the state of node001 being UP and that of node002 being DOWN:

```
[root@bright82 ~]# cmsh -c "device status"
node001 .................. [   UP   ]
node002 .................. [ DOWN   ]
bright82 ................. [   UP   ]
```

In the examples, the outputs for pdsh could be as follows for the pdsh options considered:

-A: With this pdsh option an attempt is made to run the command on all nodes, regardless of the node state:

Example
```
[root@bright82 ~]# pdsh -A hostname
node001: node001
node002: ssh: connect to host node002 port 22: No route to host
pdsh@bright82: node002: ssh exited with exit code 255
bright82: bright82
```

-v: With this option an attempt is made to run the command only on nodes nodes that are in the state UP:

Example
```
[root@bright82 ~]# pdsh -A -v hostname
node001: node001
bright82: bright82
```

-g: With this option, and using, for example, computenode as the genders query, only nodes within computenode in the /etc/genders file are considered for the command. The -v option then further ensures that the command runs only on a node in computenode that is up. In a newly-installed cluster, regular nodes are assigned to computenode by default, so the command runs on all regular nodes that are up in a newly-installed cluster:

Example
```
[root@bright82 ~]# pdsh -v -g computenode hostname
node001: node001
```

-w: This option allows a node list (man pdsh(1)) to be specified on the command line itself:

Example
```
[root@bright82 ~]# pdsh -w node00[1-2] hostname
node001: node001
node002: ssh: connect to host node002 port 22: No route to host
pdsh@bright82: node002: ssh exited with exit code 255
```
-x:  This option is the converse of -w, and excludes a node list that is specified on the command line itself:

Example

[root@bright82 ~]# pdsh -x node002 -w node00[1-2] hostname
node001: node001

The dshbak Command

The dshbak (distributed shell backend formatting filter) command is a filter that reformats pdsh output. It comes with the pdsh package.

Running dshbak with the -h option displays:

[root@bright82 ~]# dshbak -h
Usage: dshbak [OPTION]...
  -h Display this help message
  -c Coalesce identical output from hosts
  -d DIR Send output to files in DIR, one file per host
  -f With -d, force creation of DIR

Further details can be found in man dshbak(1).

For the examples in this section, it is assumed that all the nodes in the cluster are now up. That is, node002 used in the examples of the preceding section is now also up. Some examples to illustrate how dshbak works are then the following:

Without dshbak:

Example

[root@bright82 ~]# pdsh -A ls /etc/services /etc/yp.conf
bright82: /etc/services
bright82: /etc/yp.conf
node001: /etc/services
node001: ls: cannot access /etc/yp.conf: No such file or directory
pdsh@bright82: node001: ssh exited with exit code 2
node002: /etc/services
node002: /etc/yp.conf

With dshbak, with no dshbak options:

Example

[root@bright82 ~]# pdsh -A ls /etc/services /etc/yp.conf | dshbak
node001: ls: cannot access /etc/yp.conf: No such file or directory
pdsh@bright82: node001: ssh exited with exit code 2

----------------
bright82
----------------
/etc/services
/etc/yp.conf
----------------
node001
----------------
/etc/services
----------------
node002
13.1 Parallel Shells: `pdsh` And `pexec`

```
/etc/services
/etc/yp.conf
[root@bright82 ~]
```

With `dshbak`, with the `c` (coalesce) option:

**Example**

```
[root@bright82 ~]# pdsh -A ls /etc/services /etc/yp.conf | dshbak -c
node001: ls: cannot access /etc/yp.conf: No such file or directory
pdsh@bright82: node001: ssh exited with exit code 2
-----------------
node002,bright82
-----------------
/etc/services
/etc/yp.conf
```

The `dshbak` utility is useful for creating human-friendly output in clusters with larger numbers of nodes.

### 13.1.2 `pexec` In `cmsh`

In `cmsh`, the `pexec` command is run from device mode:

**Example**

```
[bright82->device]% pexec -n node001,node002 "cd ; ls"
```

```
[node001] :
anaconda-ks.cfg
install.log
install.log.syslog
```

```
[node002] :
anaconda-ks.cfg
install.log
install.log.syslog
```

### 13.1.3 `pexec` In Bright View

In Bright View, `pexec` is hidden, but executed in a GUI wrapper, using the clickpath `Cluster→Run` command.

For large numbers of nodes, rendering the output into the node subpanes (little boxes) can take a long time. To improve the Bright View experience, selecting the `Single text view` icon instead of the `Grouped` view icon speeds up the rendering significantly, but at the cost of removing the borders of the subpanes.

Ticking the `Join output` checkbox places output that is the same for particular nodes, into the same subpane.

Running parallel shell commands from `cmsh` instead of in Bright View is faster in most cases, due to less graphics rendering overhead.

© Bright Computing, Inc.
13.1.4 Using The -j|--join Option Of pexec In cmsh

The output of the pexec command by default can come out in a sequence depending on node response time. To make it more useful for an administrator, order can be imposed on the output. Checking consistency across nodes is then trivial. For example, to see if all nodes have the same mounts on a cluster with 10 nodes, of which node002 is down:

Example

[bright82->device]% pexec -j -c default "mount|sort"
Nodes down: node002
Node down
[...]
/dev/hda1 on / type ext3 (rw, noatime, nodiratime)
/dev/hda2 on /var type ext3 (rw, noatime, nodiratime)
/dev/hda3 on /tmp type ext3 (rw, nosuid, nodev, noatime, nodiratime)
/dev/hda6 on /local type ext3 (rw, noatime, nodiratime)
master:/cm/shared on /cm/shared type nfs
  (rw, rsize=32768, wsize=32768, hard, intr, addr=10.141.255.254)
master:/home on /home type nfs
  (rw, rsize=32768, wsize=32768, hard, intr, addr=10.141.255.254)
none on /dev/pts type devpts (rw, gid=5, mode=620)
none on /dev/shm type tmpfs (rw)
none on /proc/sys/fs/binfmt_misc type binfmt_misc (rw)
none on /proc type proc (rw, nosuid)
none on /sys type sysfs (rw)

Here, the -j option joins up identical fields (like the standard unix text utility, join). Additional order is imposed by sorting the output of each mount command within bash before the -j option operates from cmsh. The -c option executes the command on the default category of nodes.

13.1.5 Other Parallel Commands

Besides pexec, CMDaemon has several other parallel commands:

**pkill**: parallel kill

Synopsis:

pkill [OPTIONS] <tracker> [ <tracker> ... ]

**plist**: List the parallel commands that are currently running, with their tracker ID

Synopsis:

plist

**pping**: ping nodes in parallel

Synopsis:

pping [OPTIONS]

**pwait**: wait for parallel commands that are running to complete

Synopsis:

pwait [OPTIONS] <tracker> [ <tracker> ... ]

Details on these parallel commands, including examples, can be seen by executing the help command within the device mode of cmsh for a parallel command, <pcommand>, as follows:
13.2 Getting Support With Cluster Manager Issues

Bright Cluster Manager is constantly undergoing development. While the result is a robust and well-designed cluster manager, it is possible that the administrator may run into a feature that requires technical help from Bright Computing or Bright Computing’s resellers. This section describes how to get support for such issues.

13.2.1 Support Via E-mail

If the reseller from whom Bright Cluster Manager was bought offers direct support, then the reseller should be contacted.

Otherwise the primary means of support is e-mail. The e-mail exchanges begin via the Bright website. The website allows the administrator to submit a support request via a web form, and opens up a trouble ticket.

- A useful indirect starting point that can be used for all technical support is at:

  https://support.brightcomputing.com

- Cluster administrators that have a Bright customer portal account can click on the Request Support link in the navigation menu on the left-hand column of the portal page. A direct URL for this is:


- Cluster administrators that do not have a Bright customer portal account can use the web form at:

  https://customer.brightcomputing.com/support?p=submitSupportRequest

The correct, up-to-date, product key (section 4.3.2 of the Installation Manual) for the cluster should be entered in the web form to allow tickets to be dealt with smoothly. If the product key for the cluster has been registered (section 4.3 of the Installation Manual), then all tickets associated with the key can be viewed after logging in.

When opening up the ticket, it is a good idea to try to use a clear subject header, since that is used as part of a reference tag as the ticket progresses. Also helpful is a good description of the issue. The followup communication for this ticket goes via standard e-mail.

Putting obviously different problems into one ticket results in a slower resolution. Issues that are clearly independent of each other should be submitted as separate tickets. This ensures the quickest processing by the support team.

The document “How To Ask Questions The Smart Way” at http://www.catb.org/esr/faqs.smart-questions.html is quite readable, and explains how to submit a good technical query on online forums. While the document is not entirely relevant to the support provided by Bright Computing, it is true that many of the principles covered there are also useful when sending a query to Bright Computing when trying to resolve an issue. A read through the document may therefore be helpful before submitting a query to Bright Computing.

If the issue appears to be a bug, then a bug report should be submitted. It is helpful to include as many details as possible to ensure the development team is able to reproduce the apparent bug. The policy at Bright Computing is to welcome such reports, to provide feedback to the reporter, and to resolve the problem.

© Bright Computing, Inc.
If there is apparently no response from Bright Computing over e-mail, checking the spam folder is advised. Mail providers have been known to flag mails as spam, even in mid-thread. Users in China should also be aware that mails from Bright Computing may be deleted by accident by the Golden Shield Project (the “Great Firewall”) operated by the Chinese Ministry of Public Security.

As a supplement to e-mail support, Bright Computing also provides the cm-diagnose (section 13.2.2) and the request-remote-assistance (section 13.2.3) utilities to help resolve issues. The use of third-party screen-sharing utilities (section 13.2.4) is also possible.

13.2.2 Reporting Cluster Manager Diagnostics With cm-diagnose

The diagnostic utility cm-diagnose is run from the head node and gathers data on the system that may be of use in diagnosing issues. To view its options, capabilities, and defaults, it can be run as “cm-diagnose --help”. For particular issues it may be helpful to change some of the default values to gather data in a more targeted way.

When run without any options, it runs interactively, and allows the administrator to send the resultant diagnostics file to Bright Computing directly. The output of a cm-diagnose session looks something like the following (the output has been made less verbose for easy viewing):

Example

```
[root@bright82 ~]# cm-diagnose
To be able to contact you about the issue, please provide your e-mail address (or support ticket number, if applicable):
franknfurter@example.com

To diagnose any issue, it would help if you could provide a description about the issue at hand.
Do you want to enter such description? [Y/n]

End input with ctrl-d
I tried X, Y, and Z on the S2464 motherboard. When that didn’t work, I tried A, B, and C, but the florbish is grommicking.
Thank you.

If issues are suspected to be in the cmdaemon process, a gdb trace of that process is useful.
In general such a trace is only needed if Bright Support asks for this.
Do you want to create a gdb trace of the running CMDaemon? [y/N]

Proceed to collect information? [Y/n]

Processing master
  Processing commands
  Processing file contents
  Processing large files and log files
Collecting process information for CMDaemon
  Executing CMSH commands
  Finished executing CMSH commands

Processing default-image
  Processing commands
  Processing file contents

Creating log file: /root/bright82_609.tar.gz

Cleaning up
```

© Bright Computing, Inc.
13.2 Getting Support With Cluster Manager Issues

Automatically submit diagnostics to http://support.brightcomputing.com/cm-diagnose/ ? [Y/n] y

Uploaded file: bright82_609.tar.gz
Remove log file (/root/bright82_609.tar.gz)? [y/N] y

[58x789]13.2.3 Requesting Remote Support With request-remote-assistance

The life-cycle of solving a ticket begins with opening a ticket, and then establishing that both the cluster administrator and the support engineer have a basic grasp of the issue at hand. This is best done via an e-mail exchange.

From this stage onward there are many possible paths. The support engineer may offer a solution, or ask for more details, or may ask for some tests to be run. Most of the time e-mail remains the most efficient way to troubleshoot an issue.

However at times it may be more appropriate for the cluster administrator to allow remote support from the Bright Computing support engineer in order to resolve the issue. The support engineer may in that case suggest that the request-remote-assistance utility be run.

The request-remote-assistance utility allows a Bright Computing engineer to securely tunnel into the cluster, often without a change in firewall or ssh settings of the cluster.

With request-remote-assistance:

- It must be allowed to access the www and ssh ports of Bright Computing’s internet servers.

- For some problems, the engineer may wish to power cycle a node. In that case, indicating what node the engineer can power cycle should be added to the option for entering additional information.

- Administrators familiar with screen may wish to run it within a screen session and detach it so that they can resume the session from another machine. A very short reminder of the basics of how to run a screen session is: run the screen command, then run the request-remote-assistance command, then ctrl-a d to detach. To resume, run screen -r, and exit to exit.

The request-remote-assistance command itself is run as follows:

Example

[root@bright82 ~]# request-remote-assistance

This tool helps securely set up a temporary ssh tunnel to sandbox.brightcomputing.com.

Allow a Bright Computing engineer ssh access to the cluster? [Y/n]

Enter additional information for Bright Computing (eg: related ticket number, problem description)? [Y/n]

End input with ctrl-d
Ticket 1337 - the florbish is grommicking

Thank you.

Added temporary Bright public key.

© Bright Computing, Inc.
After the administrator has responded to the Enter additional information... entry, and has typed in the `ctrl-d`, the utility tries to establish the connection. The screen clears, and the secure tunnel opens up, displaying the following notice:

REMOTE ASSISTANCE REQUEST
########################################################
A connection has been opened to Bright Computing Support.
Closing this window will terminate the remote assistance session.

Hostname: bright82.NOFQDN
Connected on port: 7000

ctrl-c to terminate this session

Bright Computing support automatically receives an e-mail alert that an engineer can now securely tunnel into the cluster. The session activity is not explicitly visible to the administrator. Whether an engineer is logged in can be viewed with the `w` command, which shows a user running the ssh tunnel, and—if the engineer is logged in—another user session, other than whatever sessions the administrator may be running:

Example

```
[root@bright82 ~]# w
13:35:00 up 97 days, 17 min, 2 users, load average: 0.28, 0.50, 0.52
USER TTY FROM LOGIN@ IDLE JCPU PCPU WHAT
root pts/0 10.2.37.101 12:24 1:10m 0.14s 0.05s ssh -q -R :7013:127.0.0.1:22\remote@sandbox.brightcomputing.com bright82
root pts/1 localhost.locald 12:54 4.00s 0.03s 0.03s -bash
```

When the engineer has ended the session, the administrator may remove the secure tunnel with a `ctrl-c`, and the display then shows:

Tunnel to sandbox.brightcomputing.com terminated.
Removed temporary Bright public key.

The preceding tunnel termination output may also show up automatically, without a `ctrl-c` from the administrator, within seconds after the connection session starts. In that case, it typically means that a firewall is blocking access to SSH and WWW to Bright Computing’s internet servers.

13.2.4 Requesting Remote Support With A Shared Screen Utility

If the request-remote-assistance utility (section 13.2.3) is restricted or otherwise disallowed by the site policy, then it may be permitted to allow a Bright support engineer to access the site using a third party shared screen utility that the administrator is comfortable with. In that case, possible options include:

- Zoom (preferred)
- WebEx
- join.me

Other screensharing options may also be possible. The support engineer and the administrator need to agree upon the option to be used, and decide upon a time (and time zone) for the shared screen session.
13.3 Backups

13.3.1 Cluster Installation Backup

Bright Cluster Manager does not include facilities to create backups of a cluster installation. The cluster administrator is responsible for deciding on the best way to back up the cluster, out of the many possible choices.

A backup method is strongly recommended, and checking that restoration from backup actually works is also strongly recommended.

One option that may be appropriate for some cases is simply cloning the head node. A clone can be created by PXE booting the new head node, and following the procedure in section 14.4.8.

When setting up a backup mechanism, it is recommended that the full filesystem of the head node (i.e. including all software images) is backed up. Unless the regular node hard drives are used to store important data, it is not necessary to back them up.

If no backup infrastructure is already in place at the cluster site, the following open source (GPL) software packages may be used to maintain regular backups:

- **Bacula**: Bacula is a mature network based backup program that can be used to backup to a remote storage location. If desired, it is also possible to use Bacula on nodes to back up relevant data that is stored on the local hard drives. More information is available at http://www.bacula.org

  Bacula requires ports 9101-9103 to be accessible on the head node. Including the following lines in the Shorewall rules file for the head node allows access via those ports from an IP address of 93.184.216.34 on the external network:

  ```
  ACCEPT net:93.184.216.34 fw tcp 9101
  ACCEPT net:93.184.216.34 fw tcp 9102
  ACCEPT net:93.184.216.34 fw tcp 9103
  ```

  The Shorewall service should then be restarted to enforce the added rules.

- **rsnapshot**: rsnapshot allows periodic incremental filesystem snapshots to be written to a local or remote filesystem. Despite its simplicity, it can be a very effective tool to maintain frequent backups of a system. More information is available at http://www.rsnapshot.org.

  Rsnapshot requires access to port 22 on the head node.

13.3.2 Local Database And Data Backups And Restoration

The CMDaemon database is stored in the MySQL cmdaemon database, and contains most of the stored settings of the cluster.

Monitoring data values are stored as binaries in the filesystem, under /var/spool/cmd/monitoring.

The administrator is expected to run a regular backup mechanism for the cluster to allow restores of all files from a recent snapshot. As an additional, separate, convenience:

- For the CMDaemon database, the entire database is also backed up nightly on the cluster filesystem itself (“local rotating backup”) for the last 7 days.

- For the monitoring data, the raw data records are not backed up locally, since these can get very large. However, the configuration of the monitoring data, which is stored in the CMDaemon database, is backed up for the last 7 days too.

© Bright Computing, Inc.
Database Corruption Messages And Repairs
A corrupted MySQL database is commonly caused by an improper shutdown of the node. To deal with this, when starting up, MySQL checks itself for corrupted tables, and tries to repair any such by itself. Detected corruption causes an event notice to be sent to cmsh or Bright View.

When there is database corruption, InfoMessages in the /var/log/cmdaemon log may mention:

- “Unexpected eof found” in association with a table in the database,
- “can’t find file” when referring to an entire missing table,
- locked tables,
- error numbers from table handlers,
- “Error while executing” a command.

If a basic repair is to be carried out on a database, CMDaemon should first be stopped.

Example

[root@headnode ~]# service cmd stop
[root@headnode ~]# myisamchk --recover /var/lib/mysql/mysql/user.MYI
[root@headnode ~]# service cmd start

If basic repair fails, more extreme repair options—man myisamchk(1) suggests what—can then be tried out.

Another example: If CMDaemon is unable to start up due to a corrupted database, then messages in the /var/log/cmdaemon file might show something like:

Example

Oct 11 15:48:20 solaris CMDaemon: Info: Sending reconnect command to all
nodes which were up before master went down ...

Here it is CMDaemon’s “Database corruption” message that the administrator should be aware of, and which suggests database repairs are required for the CMDaemon database. The severity of the corruption, in this case not even allowing CMDaemon to start up, may mean a restoration from backup is needed. How to restore from backup is explained in the next section.

Restoring From The Local Backup
If the MySQL database repair tools of the previous section do not fix the problem, then, for a failover configuration, the dbreclone option (section 14.4.2) should normally provide a CMDaemon and Slurm database that is current. The dbreclone option does not clone the monitoring database.

Cloning the monitoring database: The cm-clone-monitoring-db.sh helper script that comes with CMDaemon can be used to clone the monitoring database.
Cloning extra databases: The file /cm/local/apps/cluster-tools/ha/conf/extradbcclone.xml template can be used as a template to create a file extradbcclone.xml in the same directory. The extradbcclone.xml file can then be used to define additional databases to be cloned. Running the /cm/local/apps/cmd/scripts/cm-update-mycnf script then updates /etc/my.cnf. The database can then be cloned with this new MySQL configuration by running

```
cmha dbreclone <passive>
```

where <passive> is the hostname of the passive headnode.

If the head node is not part of a failover configuration, then a restoration from local backup can be done. The local backup directory is /var/spool/cmd/backup, with contents that look like (some text elided):

Example

```
[root@solaris ~]# cd /var/spool/cmd/backup/
[root@solaris backup]# ls -l
total 280
... 
-rw------- 1 root root 33804 Oct 10 04:02 backup-Mon.sql.gz
-rw------- 1 root root 33805 Oct 9 04:02 backup-Sun.sql.gz
-rw------- 1 root root 33805 Oct 11 04:02 backup-Tue.sql.gz
... 
```

The CMDaemon database snapshots are stored as backup-<day of week>.sql.gz In the example, the latest backup available in the listing for CMDaemon turns out to be backup-Tue.sql.gz.

The latest backup can then be ungzipped and piped into the MySQL database for the user cmdaemon. The password, <password>, can be retrieved from /cm/local/apps/cmd/etc/cmd.conf, where it is configured in the DBPass directive (Appendix C).

Example

```
gunzip backup-Tue.sql.gz
service cmd stop #(just to make sure)
mysql -ucmdaemon -p<password> cmdaemon < backup-Tue.sql
```

Running “service cmd start” should have CMDaemon running again, this time with a restored database from the time the snapshot was taken. That means, that any changes that were done to the cluster manager after the time the snapshot was taken are no longer implemented.

Monitoring data values are not kept in a database, but in files (section 13.8).

13.4 Revision Control For Images

Bright Cluster Manager 7 introduced support for the implementations of Btrfs provided by the distributions. Btrfs makes it possible to carry out revision control for images efficiently.

13.4.1 Btrfs: The Concept And Why It Works Well In Revision Control For Images

Btrfs, often pronounced “butter FS”, is a Linux implementation of a copy-on-write (COW) filesystem.

A COW design for a filesystem follows the principle that, when blocks of old data are to be modified, then the new data blocks are written in a new location (the COW action), leaving the old, now superseded, copy of the data blocks still in place. Metadata is written to keep track of the event so that, for example, the new data blocks can be used seamlessly with the contiguous old data blocks that have not been superseded.

© Bright Computing, Inc.
This is in contrast to the simple overwriting of old data that a non-COW filesystem such as Ext3fs carries out.

A result of the COW design means that the old data can still be accessed with the right tools, and that rollback and modification become a natural possible feature.

"Cheap" revision control is thus possible.

Revision control for the filesystem is the idea that changes in the file system are tracked and can be rolled back as needed. "Cheap" here means that COW makes tracking changes convenient, take up very little space, and quick. For an administrator of Bright Cluster Manager, cheap revision control is interesting for the purpose of managing software images.

This is because for a non-COW filesystem such as Ext3fs, image variations take a large amount of space, even if the variation in the filesystem inside the image is very little. On the other hand, image variations in a COW filesystem such as Btrfs take up near-minimum space.

Thus, for example, the technique of using initialize and finalize scripts to generate such image variations on the fly (section 3.15.4) in order to save space, can be avoided by using a Btrfs partition to save the full image variations instead.

"Expensive" revision control on non-COW filesystems is also possible. It is merely not recommended, since each disk image takes up completely new blocks, and hence uses up more space. The administrator will then have to consider that the filesystem may become full much earlier. The degree of restraint on revision control caused by this, as well as the extra consumption of resources, means that revision control on non-COW filesystems is best implemented on test clusters only, rather than on production clusters.

### 13.4.2 Btrfs Availability And Distribution Support

Btrfs has been part of the Linux kernel since kernel 2.6.29-rc1. Depending on which Linux distribution is being used on a cluster, it may or may not be a good idea to use Btrfs in a production environment, as in the worst case it could lead to data loss.

Btrfs is available from Red Hat as a technology preview release in versions 6 and 7 for now. That means it is not officially supported by Red Hat. The Btrfs version used in the 2.6 kernel series, used by RHEL6.x is quite old, which means that it may not be fully stable. Btrfs users who would like significantly more features and stability will prefer the 3.x kernel series, used by RHEL7.x. Btrfs has been deprecated since RHEL7.4.

Btrfs is also available from SUSE. SLES12 uses a 3.12 series kernel, which has a fully stable and supported implementation of Btrfs.

While Bright Computing has not experienced any problems with storing software images on Btrfs using RHEL6, it is highly advisable to keep backups of important software images on a non-Btrfs filesystems when Btrfs is used on these Linux distributions.

An issue with using `cm-clone-install` with Btrfs is described on page 607.

### 13.4.3 Installing Btrfs To Work With Revision Control Of Images In Bright Cluster Manager

#### Installation Of btrfs-progs

To install a Btrfs filesystem, the `btrfs-progs` packages must be installed from the distribution repository first (some lines elided):

**Example**

```
[root@bright82 ~]# yum install btrfs-progs
...
Resolving Dependencies
---> Running transaction check
---> Package btrfs-progs.x86_64 0:0.20-0.2.git91d9e9c.el6 will be installed
...
Total download size: 297 k
Installed size: 2.4 M
```
Complete!

**Creating A Btrfs Filesystem**

The original images directory can be moved aside first, and a new images directory created to serve as a future mount point for Btrfs:

**Example**

```
[root@bright82 ~]# cd /cm/
[root@bright82 cm]# mv images images2
[root@bright82 cm]# mkdir images
```

A block device can be formatted as a Btrfs filesystem in the usual way by using the `mkfs.btrfs` command on a partition, and then mounted to the new images directory:

**Example**

```
[root@bright82 cm]# mkfs.btrfs /dev/sdc1
[root@bright82 cm]# mount /dev/sdc1 /cm/images
```

If there is no spare block device, then, alternatively, a file with zeroed data can be created, formatted as a Btrfs filesystem, and mounted as a loop device like this:

**Example**

```
[root@bright82 cm]# dd if=/dev/zero of=butter.img bs=1G count=20
20+0 records in
20+0 records out
21474836480 bytes (21 GB) copied, 916.415 s, 23.4 MB/s
[root@bright82 cm]# mkfs.btrfs butter.img
```

**WARNING!** - Btrfs Btrfs v0.20-rc1 IS EXPERIMENTAL

**WARNING!** - see http://btrfs.wiki.kernel.org before using

```
fs created label (null) on butter.img
   nodesize 4096 leafsize 4096 sectorsize 4096 size 20.00GB
Btrfs Btrfs v0.20-rc1
[root@bright82 cm]# mount -t btrfs butter.img images -o loop
[root@bright82 cm]# mount
... 
/cm/butter.img on /cm/images type btrfs (rw,loop=/dev/loop0)
```

**Migrating Images With cm-migrate-images**

The entries inside the `/cm/images/` directory are the software images (file trees) used to provision nodes.

Revision tracking of images in a Btrfs filesystem can be done by making the directory of a specific image a *subvolume*. Subvolumes in Btrfs are an extension of standard unix directories with versioning. The files in a subvolume are tracked with special internal Btrfs markers.

To have this kind of version tracking of images work, image migration cannot simply be done with a `cp` or `mv` command. That is, moving images from the `images2` directory in the traditional filesystem, over to images in the `images` directory in the Btrfs filesystem command with the standard `cp` or `mv` command is not appropriate. This is because the images are to be tracked once they are in the Btrfs system, and are therefore not standard files any more, but instead extended, and made into subvolumes.

The migration for Bright Cluster Manager software images can be carried out with the utility `cm-migrate-image`, which has the usage:

```
cm-migrate-image <path to old image> <path to new image>
```

where the old image is a traditional directory, and the new image is a subvolume.

© Bright Computing, Inc.
Example

[root@bright82 cm]# cm-migrate-image /cm/images2/default-image /cm/images/default-image

The default-image directory, or more exactly, subvolume, must not exist in the Btrfs filesystem before the migration. The subvolume is created only for the image basename that is migrated, which is default-image in the preceding example.

In the OS, the btrfs utility is used to manage Btrfs. Details on what it can do can be seen in the btrfs(8) man page. The state of the loop filesystem can be seen, for example, with:

Example

[root@bright82 cm]# btrfs filesystem show /dev/loop0
Label: none  uuid: 6f94de75-2e8d-45d2-886b-f87326b73474
   Total devices 1 FS bytes used 3.42GB
devid 1 size 20.00GB used 6.04GB path /dev/loop0

Btrfs Btrfs v0.20-rc1
[root@bright82 cm]# btrfs subvolume list /cm/images
ID 257 gen 482 top level 5 path default-image

The filesystem can be modified as well as merely viewed with btrfs. However, instead of using the utility to modify image revisions directly, it is recommended that the administrator use Bright Cluster Manager to manage image version tracking, since the necessary functionality has been integrated into cmsh and Bright View.

13.4.4 Using cmsh For Revision Control Of Images

Revision Control Of Images Within softwareimage Mode

The following commands and extensions can be used for revision control in the softwareimage mode of cmsh:

- **newrevision** `<parent software image name> "textual description"`
  Creates a new revision of a specified software image. For that new revision, a revision number is automatically generated and saved along with time and date. The new revision receives a name of the form:

  `<parent software image name>@<revision number>`

  A new Btrfs subvolume:

  `/cm/images/<parent software image name>-<revision number>`

  is created automatically for the revision. From now on, this revision is a self-contained software image and can be used as such.

- **revisions** `-a|--all` `<parent software image name>`
  Lists all revisions of specified parent software image in the order they were created, and associates the revision with the revision number under the header `ID`. The option `-a|--all` also lists revisions that have been removed.

- **list** `-r|--revisions`
  The option `-r|--revisions` has been added to the `list` command. It lists all revisions with their name, path, and kernel version. A parent image is the one at the head of the list of revisions, and does not have `@<revision number>` in its name.

© Bright Computing, Inc.
- **setparent** *<parent software image name> <revision name>*

  Sets a revision as a new parent. The action first saves the image directory of the current, possibly altered, parent directory or subvolume, and then attempts to copy or snapshot the directory or subvolume of the revision into the directory or subvolume of the parent. If the attempt fails, then it tries to revert all the changes in order to get the directory or subvolume of the parent back to the state it was in before the attempt.

- **remove** *[-a|--all] [-d|--data] <parent software image name>*

  Running remove without options removes the parent software image. The option `-a|--all` removes the parent and all its revisions. The option `-d|--data` removes the actual data. To run the remove command, any images being removed should not be in use.

### Revision Control Of Images Within category Mode

The category mode of cmsh also supports revision control.

Revision control can function in 3 kinds of ways when set for the `softwareimage` property at category level.

To explain the settings, an example can be prepared as follows: It assumes a newly-installed cluster with Btrfs configured and `/cm/images` migrated as explained in section 13.4.3. The cluster has a default software image `default-image`, and two revisions of the parent image are made from it. These are automatically given the paths `default-image@1` and `default-image@2`. A category called `storage` is then created:

**Example**

```plaintext
[bright82->softwareimage]% newrevision default-image "some changes"
[bright82->softwareimage]% newrevision default-image "more changes"
[bright82->softwareimage]% revisions default-image
```

<table>
<thead>
<tr>
<th>ID</th>
<th>Date</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Wed, 07 May 2014 05:28:06 PDT</td>
<td>some changes</td>
</tr>
<tr>
<td>2</td>
<td>Wed, 07 May 2014 05:28:55 PDT</td>
<td>more changes</td>
</tr>
</tbody>
</table>

```plaintext
[bright82->softwareimage]% list -r
```

<table>
<thead>
<tr>
<th>Name (key)</th>
<th>Path</th>
<th>Kernel version</th>
</tr>
</thead>
<tbody>
<tr>
<td>default-image</td>
<td>/cm/images/default-image</td>
<td>2.6.32-431.11.2.el6.x86_64</td>
</tr>
<tr>
<td>default-image@1</td>
<td>/cm/images/default-image-1</td>
<td>2.6.32-431.11.2.el6.x86_64</td>
</tr>
<tr>
<td>default-image@2</td>
<td>/cm/images/default-image-2</td>
<td>2.6.32-431.11.2.el6.x86_64</td>
</tr>
</tbody>
</table>

```plaintext
[bright82->softwareimage]% category add storage; commit
```

With the cluster set up like that, the 3 kinds of revision control functionalities in category mode can be explained as follows:

1. **Category revision control functionality is defined as unset**

   If the administrator sets the `softwareimage` property for the category to an image without any revision tags:

   **Example**

   ```plaintext
   [bright82->category]% set storage softwareimage default-image
   ```

   then nodes in the `storage` category take no notice of the revision setting for the image set at category level.
2. **Category revision control sets a specified revision as default**

If the administrator sets the `softwareimage` property for the category to a specific image, with a revision tag, such as `default-image@1`:

**Example**

```
[bright82->category]% set storage softwareimage default-image@1
```

then nodes in the storage category use the image `default-image@1` as their image if nothing is set at node level.

3. **Category revision control sets the latest available revision by default**

If the administrator sets the `softwareimage` property for the category to a parent image, but tagged with the reserved keyword tag `latest`:

**Example**

```
[bright82->category]% set storage softwareimage default-image@latest
```

then nodes in the storage category use the image `default-image@2` if nothing is set at node level. If a new revision of `default-image` is created later on, with a later tag (`@3`, `@4`, `@5`,...) then the property takes the new value for the revision, so that nodes in the category will use the new revision as their image.

---

### Revision Control For Images—An Example Session

This section uses a session to illustrate how image revision control is commonly used, with commentary along the way. It assumes a newly installed cluster with Btrfs configured and `/cm/images` migrated as explained in section 13.4.3.

First, a revision of the image is made to save its initial state:

```
[bright82->softwareimage]% newrevision default-image "Initial state"
```

A new image `default-image@1` is automatically created with a path `/cm/images/default-image-1`. The path is also a subvolume, since it is on a Btrfs partition.

The administrator then makes some modifications to the parent image `/cm/images/default-image`, which can be regarded as a “trunk” revision, for those familiar with SVN or similar revision control systems. For example, the administrator could install some new packages, edit some configuration files, and so on. When finished, a new revision is created by the administrator:

```
[bright82->softwareimage]% newrevision default-image "Some modifications"
```

This image is then automatically called `default-image@2` and has the path `/cm/images/default-image-2`. If the administrator then wants to test the latest revision on nodes in the `default` category, then this new image can be set at category level, without having to specify it for every node in that category individually:

```
[bright82->category]% set default softwareimage default-image@2
```

At this point, the content of `default-image` is identical to `default-image@2`. But changes done in `default-image` will not affect what is stored in revision `default-image@2`.

After continuing on with the parent image based on the revision `default-image@2`, the administrator might conclude that the modifications tried are incorrect or not wanted. In that case, the administrator can roll back the state of the parent image `default-image` back to the state that was previously saved as the revision `default-image@1`:

© Bright Computing, Inc.
The administrator can thus continue experimenting, making new revisions, and trying them out by setting the `softwareimage` property of a category accordingly. Previously created revisions `default-image@1` and `default-image@2` will not be affected by these changes. If the administrator would like to completely purge a specific unused revision, such as `default-image@2` for example, then it can be done with the `-d|--data` option:

```
[bright82->softwareimage]% remove -d default-image@2
```

The `-d` does a forced recursive removal of the `default-image-2` directory, while a plain remove without the `-d` option would simply remove the object from CMDaemon, but leave `default-image-2` alone. This CMDaemon behavior is not unique for Btrfs—it is true for traditional filesystems too. It is however usually very wasteful of storage to do this with non-COW systems.

## 13.5 BIOS Configuration And Updates

A number of tools can be used to configure and update the BIOS of nodes.

### 13.5.1 Updating BIOS

Upgrading the BIOS to a new version involves using the DOS tools that were supplied with the BIOS. The flash tool and the BIOS image must be copied to a DOS image. The file `autoexec.bat` should be altered to invoke the flash utility with the correct parameters. In case of doubt, it can be useful to boot the DOS image and invoke the BIOS flash tool manually. Once the correct parameters have been determined, they can be added to the `autoexec.bat`.

After a BIOS upgrade, the contents of the NVRAM may no longer represent a valid BIOS configuration because different BIOS versions may store a configuration in different formats. It is therefore recommended to also write updated NVRAM settings immediately after flashing a BIOS image.

The next section describes how to boot the DOS image.

### 13.5.2 Booting DOS Image

To boot the DOS image over the network, it first needs to be copied to software image’s `/boot` directory, and must be world-readable.

**Example**

```
cp flash.img /cm/images/default-image/boot/bios/flash.img
chmod 644 /cm/images/default-image/boot/bios/flash.img
```

An entry is added to the PXE boot menu to allow the DOS image to be selected. This can easily be achieved by modifying the contents of `/cm/images/default-image/boot/bios/menu.conf`, which is by default included automatically in the PXE menu. By default, one entry example is included in the PXE menu, which is however invisible as a result of the `MENU HIDE` option. Removing the `MENU HIDE` line will make the BIOS flash option selectable. Optionally the `LABEL` and `MENU LABEL` may be set to an appropriate description.

The option `MENU DEFAULT` may be added to make the BIOS flash image the default boot option. This is convenient when flashing the BIOS of many nodes.

**Example**

```
LABEL FLASHBIOS
  KERNEL memdisk
  APPEND initrd=flash.img
  MENU LABEL "Flash BIOS"
  # MENU HIDE
  MENU DEFAULT
```

© Bright Computing, Inc.
The bios/menu.conf file may contain multiple entries corresponding to several DOS images to allow for flashing of multiple BIOS versions or configurations.

13.6 Hardware Match Check With The `hardware-profile` Data Producer

Often a large number of identical nodes may be added to a cluster. In such a case it is a good practice to check that the hardware matches what is expected. This can be done easily as follows:

1. The new nodes, say node129 to node255, are committed to a newly-created category `newbunch` as follows (output truncated):

   ```
   [root@bright82 ~]# cmsh -c "category add newbunch; commit"
   [root@bright82 ~]# for i in {129..255}
   > do
   > cmsh -c "device; set node00$i category newbunch; commit"
   > done
   Successfully committed 1 Devices
   Successfully committed 1 Devices
   ```

   For larger clusters, reducing the time wasted in opening up `cmsh` can be done by replacing the offending `for` loop with a construction that is more elegant, but less familiar to most people:

   ```
   [root@bright82 ~]# for ((i=129; i<=255; i++)) do echo "
   > device
   > set node00$i category newbunch
   > commit"; done | cmsh
   ```

2. The hardware profile of one of the new nodes, say node129, is saved into the category `newbunch`. This is done using the `node-hardware-profile` health check script:

   **Example**

   ```
   [root@bright82 ~]# /cm/local/apps/cmd/scripts/healthchecks/node-hardware-profile -n node129 -s newbunch
   ```

   The profile is intended to be the reference hardware against which all the other nodes should match, and is saved under the directory `/cm/shared/apps/cmd/hardware-profiles/`, and further under the directory name specified by the `-s` option, which in this case is `newbunch`.

3. The `hardware-profile` data producer (section 12.2.10) can then be enabled, and the sampling frequency set as follows:

   ```
   [root@bright82 ~]# cmsh
   [bright82]->monitoring->setup use hardware-profile
   [bright82->monitoring->setup[hardware-profile]]% set interval 600; set disabled no; commit
   ```

   The `hardware-profile` data producer should also be set to the category `newbunch` created in the earlier step. This can be done by creating a category group within the `nodeexecutionfilters` submode. Within that group, categories can be set for where the hardware check is to run. For the example, it is just run on one category, `newbunch`:

   ```
   [bright82->monitoring->setup[hardware-profile]]% nodeexecutionfilters
   [bright82->...-profile]->nodeexecutionfilters% add category filterhwp
   [bright82->...-profile]->nodeexecutionfilters*[/filterhwp*]% set categories newbunch
   [bright82->...-profile]->nodeexecutionfilters*[/filterhwp*]% commit
   ```
4. CMDaemon then automatically alerts the administrator if one of the nodes does not match the hardware of that category during the first automated check. In the unlikely case that the reference node is itself faulty, then that will also be obvious because all—or almost all, if more nodes are faulty—of the other nodes in that category will then be reported “faulty” during the first check.

13.7 Serial Over LAN Console Access

Direct console access to nodes is not always possible. Other possibilities to access the node are:

1. **SSH access via an ssh client.** This requires that an ssh server runs on the node and that it is accessible via the network. Access can be via one of the following options:
   - a regular SSH client, run from a bash shell
   - via an ssh command run from the device mode of cmsh
   - via an ssh terminal launched from Bright View via the clickpath: Devices → Nodes → node → Connect → ssh.

2. **Remote shell via CMDaemon.** This is possible if CMDaemon is running on the node and accessible via Bright View or cmsh.
   - An interactive root shell session can be started up on a node via the clickpath: Devices → Nodes → node → Connect → Root shell.
     This session is connected to the node via CMDaemon, and runs bash by default.
   - For cmsh, in device mode, running the command rshell node001 launches an interactive bash session connected to node001 via CMDaemon.

3. **Connecting via a serial over LAN console.** If a serial console is configured, then a serial over LAN (SOL) console can be accessed from cmsh (rconsole).

   Item 3 in the preceding list, SOL access, is a useful low-level access method that is covered next more thoroughly with:
   - some background notes on serial over LAN console access (section 13.7.1)
   - the configuration of SOL with Bright View (section 13.7.2)
   - the configuration of SOL with cmsh (section 13.7.3)
   - the conman SOL logger and viewer (section 13.7.4)

13.7.1 Background Notes On Serial Console And SOL

Serial ports are data ports that can usually be enabled or disabled for nodes in the BIOS.

If the serial port of a node is enabled, it can be configured in the node kernel to redirect a console to the port. The serial port can thus provide what is called serial console access. That is, the console can be viewed using a terminal software such as minicom (in Linux) or Hyperterminal (in Windows) on another machine to communicate with the node via the serial port, using a null-modem serial cable. This has traditionally been used by system administrators when remote access is otherwise disabled, for example if ssh access is not possible, or if the TCP/IP network parameters are not set up right.

While traditional serial port console access as just described can be useful, it is inconvenient, because of having to set arcane serial connection parameters, use the relatively slow serial port and use a special serial cable. Serial Over LAN (SOL) is a more recent development of serial port console access, which uses well-known TCP/IP networking over a faster Ethernet port, and uses a standard Ethernet cable. SOL is thus generally more convenient than traditional serial port console access. The serial port DB-9 or DB-25 connector and its associated 16550 UART chip rarely exist on modern servers that support SOL,
but they are nonetheless usually implied to exist in the BIOS, and can be “enabled” or “disabled” there, thus enabling or disabling SOL.

SOL is a feature of the BMC (Baseboard Management Controller) for IPMI 2.0 and iLO. For DRAC, CIMC, and Redfish, SOL via IPMI is used. SOL is enabled by configuring the BMC BIOS. When enabled, data that is going to the BMC serial port is sent to the BMC LAN port. SOL clients can then process the LAN data to display the console. As far as the node kernel is concerned, the serial port is still just behaving like a serial port, so no change needs to be made in kernel configuration in doing whatever is traditionally done to configure serial connectivity. However, the console is now accessible to the administrator using the SOL client on the LAN.

SOL thus allows SOL clients on the LAN to access the Linux serial console if

1. SOL is enabled and configured in the BMC BIOS
2. the serial console is enabled and configured in the node kernel
3. the serial port is enabled and configured in the node BIOS

The BMC BIOS, node kernel, and node BIOS therefore all need to be configured to implement SOL console access.

**Background Notes: BMC BIOS Configuration**
The BMC BIOS SOL values are usually enabled and configured as a submenu or pop-up menu of the node BIOS. These settings must be manually made to match the values in Bright Cluster Manager, or vice versa.

During a factory reset of the node, it is likely that a SOL configuration in the cluster manager will no longer match the configuration on the node BIOS after the node boots. This is because the cluster manager cannot configure these. This is in contrast to the IP address and user authentication settings of the BMC (section 3.7), which the cluster manager is able to configure on reboot.

**Background Notes: Node Kernel Configuration**
Sections 13.7.2 and 13.7.3 explain how SOL access configuration is set up for the node kernel using Bright View or cmsh. SOL access configuration on the node kernel is serial access configuration on the node kernel as far as the system administrator is concerned; the only difference is that the word “serial” is replaced by “SOL” in Bright Cluster Manager’s Bright View and cmsh front ends to give a cluster perspective on the configuration.

**Background Notes: Node BIOS Configuration**
Since BIOS implementations vary, and serial port access is linked with SOL access in various ways by the BIOS designers, it is not possible to give short and precise details on how to enable and configure them. The following rules-of-thumb, if followed carefully, should allow most BMCs to be configured for SOL access with Bright Cluster Manager:

- Serial access, or remote access via serial ports, should be enabled in the BIOS, if such a setting exists.
- The node BIOS serial port settings should match the node configuration SOL settings (section 13.7.3). That means, items such as “SOL speed”, “SOL Flow Control”, and “SOL port” in the node configuration must match the equivalent in the node BIOS. Reasonable values are:
  - SOL speed: 115200bps. Higher speeds are sometimes possible, but are more likely to have problems.
  - SOL flow control: On. It is however unlikely to cause problems if flow control is off in both.
  - SOL port: COM1 (in the BIOS serial port configuration), corresponding to ttyS0 (in the node kernel serial port configuration). Alternatively, COM2, corresponding to ttyS1. Sometimes,
the BIOS configuration display indicates SOL options with options such as: “COM1 as SOL”, in which case such an option should be selected for SOL connectivity.

- Terminal type: VT100 or ANSI.

- If there is an option for BIOS console redirection after BIOS POST, it should be disabled.
- If there is an option for BIOS console redirection before or during BIOS POST, it should be enabled.
- The administrator should be aware that the BMC LAN traffic, which includes SOL traffic, can typically run over a dedicated NIC or over a shared NIC. The choice of dedicated or shared is toggled, either in the BIOS, or via a physical toggle, or both. If BMC LAN traffic is configured to run on the shared NIC, then just connecting a SOL client with an Ethernet cable to the dedicated BMC NIC port shows no console.
- The node BIOS values should manually be made to match the values in Bright Cluster Manager, or vice versa.

### 13.7.2 SOL Console Configuration With Bright View

In Bright View, SOL configuration settings can be carried out per image via the clickpath

Provisioning → Software Images → [image] → Edit → Settings

If the Enable SOL option is set to Yes then the kernel option to make the Linux serial console accessible is used after the node is rebooted.

This means that if the serial port and SOL are enabled for the node hardware, then after the node reboots the Linux serial console is accessible over the LAN via an SOL client.

If SOL is correctly configured in the BIOS and in the image, then access to the Linux serial console is possible via the minicom serial client running on the computer (from a bash shell for example), or via the rconsole serial client running in cmsh.

### 13.7.3 SOL Console Configuration And Access With cmsh

In cmsh, the serial console kernel option for a software image can be enabled within the softwareimage mode of cmsh. For the default image of default-image, this can be done as follows:

**Example**

```
[root@bright82 ~]# cmsh
[bright82]% softwareimage use default-image

[bright82->softwareimage[default-image]]% set enablesol yes
[bright82->softwareimage*[default-image*]]% commit
```

The SOL settings for a particular image can be seen with the show command:

```
[bright82->softwareimage[default-image]]% show | grep SOL
Parameter          Value
------------------- ---------------
Enable SOL         yes
SOL Flow Control   yes
SOL Port           ttyS1
SOL Speed          115200
```

Values can be adjusted if needed with the set command.

On rebooting the node, the new values are used.

To access a node via an SOL client, the node can be specified from within the device mode of cmsh, and the rconsole command run on cmsh on the head node:

**Example**
screen cleared and the following conman output is displayed:

```
conman
To exit IPMI SOL, type <ENTER> "&" "."
```

---

<ConMan> Connection to console [node001] opened.

If at this point, there is no further response in conman on the console after pressing the <ENTER> key, then there is a communication failure, probably due to a misconfigured communication parameter. This could happen, for example, if the serial port ttyS1 has been set, but the node is connected on ttyS0. Setting the value of SOL Port to ttyS0 and rebooting the node to pick up the new value, would solve that issue, so that pressing the <ENTER> key, would display the node console:

Example

Ubuntu 18.04.2 LTS node001 ttyS0

node001 login:

---

13.7.4 The conman Serial Console Logger And Viewer

In Bright Cluster Manager, the console viewer and logger service conman is used to connect to an SOL console and log the console output.

If the “Enable SOL” option in Bright View, or if the enablesol in cmsh is enabled for the software image, then the conman configuration is written out and the conman service is started.

Logging The Serial Console

The data seen at the serial console is then logged via SOL to the head node after reboot. For each node that has logging enabled, a log file is kept on the head node. For example, for node001 the log file would be at /var/log/conman/node001.log. To view the logged console output without destroying terminal settings, using less with the -R option is recommended, as in: less -R /var/log/conman/node001.log.

Using The Serial Console Interactively

Viewing quirk during boot: In contrast to the logs, the console viewer shows the initial booting stages of the node as it happens. There is however a quirk the system administrator should be aware of:

Normally the display on the physical console is a copy of the remote console. However, during boot, after the remote console has started up and been displaying the physical console for a while, the physical console display freezes. For the Linux 2.6 kernel series, the freeze occurs just before the ramdisk is run, and means that the display of the output of the launching init.d services is not seen on the physical console (figure 13.1).
The freeze is only a freeze of the display, and should not be mistaken for a system freeze. It occurs because the kernel is configured during that stage to send to only one console, and that console is the remote console. The remote console continues to display its progress (figure 13.2) during the freeze of the physical console display.
Figure 13.2: Remote Console Continues During SOL Access During Physical Console Freeze

Finally, just before login is displayed, the physical console once more (figure 13.3) starts to display what is on the remote console (figure 13.4).
Scientific Linux release 6.2 (Carbon)  
Kernel 2.6.32-220.13.1.1.el6.x86_64 on an x86_64

nodc003 login: 

Figure 13.3: Physical Console Resumes After Freeze During SOL Access

Scientific Linux release 6.2 (Carbon)  
Kernel 2.6.32-220.13.1.1.el6.x86_64 on an x86_64

nodc003 login: 

Figure 13.4: Remote Console End Display After Boot

The physical console thus misses displaying several parts of the boot progress.
Exit sequence: The `conman` console viewer session can be exited with the sequence `,`. (the last entry in the sequence being a period). Strictly speaking, the `,` sequence must actually be preceded by an `<ENTER>`.

The console buffer issue when accessing the remote console: A feature of SOL console clients is that the administrator is not presented with any text prompt from the node that is being accessed. This is useful in some cases, and can be a problem in others.

An example of the issue is the case where the administrator has already logged into the console and typed in a command in the console shell, but has no intention of pressing the `<ENTER>` key until some other tasks are first carried out. If the connection breaks at this point, then the command typed in is held in the console shell command buffer, but is not displayed when a remote serial connection is re-established to the console—the previously entered text is invisible to the client making the connection. A subsequent `<ENTER>` would then attempt to execute the command. This is why an `<ENTER>` is not sent as the last key sequence during automated SOL access, and it is left to the administrator to enter the appropriate key strokes.

To avoid commands in the console shell buffer inadvertently being run when taking over the console remotely, the administrator can start the session with a `<CTRL>-u` to clear out text in the shell before pressing `<ENTER>`.

### 13.8 Managing Raw Monitoring Data

From Bright Cluster Manager version 8.0 onwards, the raw monitoring data values are stored as binary data under `/var/spool/cmd/monitoring` instead of as binary data within MySQL or MariaDB. The reason behind this change was to significantly increase performance. The monitoring subsystem in Bright Cluster Manager was thoroughly rewritten for this change.

#### 13.8.1 Monitoring Subsystem Disk Usage With The `monitoringinfo --storage` Option

The disk usage by the monitoring subsystem can be viewed using the `monitoringinfo` command with the `--storage` option:

**Example**

```
[bright82->device]# monitoringinfo master --storage
Storage Elements Disk size Usage Free disk
----------- -------- ---------- ---------- --------
Mon::Storage::Engine 1,523 1.00 GiB 1.28% 14.1 GiB
Mon::Storage::Message 1 16.0 MiB 0.000% -
Mon::Storage::RepositoryId 1,528 47.7 KiB 100.0% -
```

The Engine component stores the raw monitoring data. It grows in 1GB increments each time its usage reaches 100%.

#### 13.8.2 Estimating The Required Size Of The Storage Device

The final size of the monitoring directory can be estimated with the Bright Cluster Manager script, `cm-bright-monitoring-usage.py`.

The size estimate assumes that there are no changes in configuration, such as enabling advanced metrics for jobs, or increasing the maximum number of labeled entities, or large numbers of running jobs.

The size estimate value is the maximum value it will take if the cluster runs forever. It is therefore an over-estimate in practice.

**Example**
13.8 Managing Raw Monitoring Data

```
[root@bright82 ~]# /cm/local/apps/cmd/scripts/monitoring/cm-bright-monitoring-usage.py
Number of used entities: 6
Number of used measurables: 231
Number of measurables: 231
Number of data producers: 95
Number of consolidators: 2

Current monitoring directory: /var/spool/cmd/monitoring
Monitoring directory size: 1.024 GB
Maximal directory size: 1.125 GB
```

### 13.8.3 Moving Monitoring Data Elsewhere

A procedure to move monitoring data from the default `/var/spool/cmd/monitoring/` directory to a new directory is as follows:

1. A new directory in which monitoring should be saved is picked. This should not be on a shared DAS or NAS device. That is because if there is an outage, then:
   - If such a DAS storage becomes unavailable at some time, then CMDaemon assumes that no monitoring data values exist, and creates an empty data file on the local storage. If the DAS storage comes back and is mounted again, then it hides the underlying files, which would lead to discontinuous values and related issues.
   - If such a NAS storage is used, then an outage of the NAS can make CMDaemon unresponsive as it waits for input and output. In addition, when CMDaemon starts with a NAS storage, and if the NAS is unavailable for some reason, then an inappropriate mount may happen as in the DAS storage case, leading to discontinuous values and related issues.

2. The `MonitoringPath` directive (page 651) is given the new directory as its value.

3. CMDaemon is stopped (`service cmd stop`).

4. The `/var/spool/cmd/monitoring/` directory is moved to the new directory.

5. CMDaemon is restarted (`service cmd start`).

### 13.8.4 Reducing Monitoring Data By Reducing Samples

Options to reduce the amount of monitoring data gathered include reducing the `Maximal age` and `Maximal samples` for data producers (section 12.4.1) to smaller, but still non-zero values. After re-initializing the monitoring database, so that existing data is removed, the values reported by the `cm-bright-monitoring-usage.py` script (section 13.8.2) then show the new storage estimates for the monitoring data.

### 13.8.5 Deleting All Monitoring Data

A procedure to delete all monitoring data from the default `/var/spool/cmd/monitoring/` directory is as follows:

1. On the active head node, the CMDaemon service can be stopped with:
   ```
   pdsh -g all service cmd stop
   ```

2. On both head nodes, the monitoring data is removed with:
   ```
   rm -rf /var/spool/cmd/monitoring/*
   ```

3. On both head nodes, the associated database tables for the CMDaemon user are cleared with a session run on each head node.
The CMDaemon database user is cmdaemon by default, but the value can be checked with a grep on the cmd.conf file:

```
[root@bright82 ~]# grep ^DBUser /cm/local/apps/cmd/etc/cmd.conf
DBUser = "cmdaemon"
```

Similarly, the password for the cmdaemon user can be found with a grep as follows:

```
[root@bright82 ~]# grep ^DBPass /cm/local/apps/cmd/etc/cmd.conf
DBPass = "slarti8813bartfahrt"
```

The monitoring measurables can then be deleted by running a session on each head node as follows:

**Example**

```
[root@bright82 ~]# mysql -ucmdaemon -p
Enter password:
Welcome to the MariaDB monitor. Commands end with ; or \g.
Your MariaDB connection id is 2909
Server version: 5.5.56-MariaDB MariaDB Server

Copyright (c) 2000, 2017, Oracle, MariaDB Corporation Ab and others.
Type 'help;' or '\h' for help. Type '\c' to clear the current input statement.

MariaDB [(none)]> use cmdaemon;
Database changed
MariaDB [cmdaemon]> truncate MonitoringMeasurables;
MariaDB [cmdaemon]> truncate MonitoringMeasurableMetrics;
MariaDB [cmdaemon]> truncate MonitoringMeasurableHealthChecks;
MariaDB [cmdaemon]> truncate MonitoringMeasurableEnums;
MariaDB [cmdaemon]> truncate EntityMeasurables;
MariaDB [cmdaemon]> truncate EnumMetricValues;
MariaDB [cmdaemon]> exit
```

4. On both head nodes, CMDaemon can then be restarted with:
   ```
   service cmd start
   ```

5. On the active head node, after the command:
   ```
   cmha status
   ```
   shows all is OK, the CMDaemon service can be started on the active node again. The OK state should be achieved in about 15 seconds.

The CMDaemon service is started with, for example:
```
pdsh -g computenode service cmd start
```

### 13.9 Node Replacement

To replace an existing node with a new node, the node information can be updated via cmsh. If the new MAC address is known, then it can set that for the node. If the MAC address is not known, then the existing entry can be cleared.

If the MAC address is not known ahead of time, then the node name for the machine should be selected when it is provisioning for the first time. The steps for a new node node031 would be as follows:
Example

[root@bright82 ~]# cmsh
[root@bright82 ~]# device use node031
  if new mac address is known, then:
  [root@bright82->device[node031] ~]# set mac <new mac address>
  else if new mac address is not known:
  [root@bright82->device[node031] ~]# clear mac
  the changed setting in either case must be committed:
  [root@bright82->device[node031] ~]# commit

If the disk is the same size as the one that is being replaced, and everything else matches up, then this should be all that needs to be done

There is more information on the node installing system in section 5.4. How to add a large number of nodes at a time efficiently is described in that section. The methods used can include the `newnodes` command of `cmsh` (page 171) and the `Nodes Identification` resource of Bright View (page 175).
14 High Availability

14.0 Introduction

14.0.1 Why Have High Availability?
In a cluster with a single head node, the head node is a single point of failure for the entire cluster. It is often unacceptable that the failure of a single machine can disrupt the daily operations of a cluster.

14.0.2 High Availability Is Possible On Head Nodes, And Also On Regular Nodes
The high availability (HA) feature of Bright Cluster Manager therefore allows clusters to be set up with two head nodes configured as a failover pair, with one member of the pair being the active head. The purpose of this design is to increase availability to beyond that provided by a single head node.

Especially with smaller clusters, it is often convenient to run all services on the head node. However, an administrator may want or need to run a service on a regular node instead. For example, a workload manager, or NFS could be run on a regular node. If a service disruption is unacceptable here, too, then HA can be configured for regular nodes too. For regular nodes, HA is done differently compared with head nodes.

By default, in this and other chapters, HA is about a head node failover configuration. When it is otherwise, then it is made explicitly clear in the manuals that it is regular node HA that is being discussed.

14.0.3 High Availability Usually Uses Shared Storage
HA is typically configured using shared storage (section 14.1.5), such as from an NFS service, which typically provides the /home directory on the active (section 14.1.1) head, and on the regular nodes.

14.0.4 Organization Of This Chapter
The remaining sections of this chapter are organized as follows:

- **HA On Head Nodes**
  - Section 14.1 describes the concepts behind HA, keeping the Bright Cluster Manager configuration in mind.
  - Section 14.2 describes the normal user-interactive way in which the Bright Cluster Manager implementation of a failover setup is configured.
  - Section 14.3 describes the implementation of the Bright Cluster Manager failover setup in a less user-interactive way, which avoids using the Ncurses dialogs of section 14.2
  - Section 14.4 describes how HA is managed with Bright Cluster Manager after it has been set up.

- **HA On Regular Nodes**

© Bright Computing, Inc.
Section 14.5 describes the concepts behind HA for regular nodes, and how to configure HA for them.

- **HA And Workload Manager Jobs**
  - Section 14.6 describes the support for workload manager job continuation during HA failover.

## 14.1 HA Concepts

### 14.1.1 Primary, Secondary, Active, Passive

**Naming:** In a cluster with an HA setup, one of the head nodes is named the *primary* head node and the other head node is named the *secondary* head node.

**Mode:** Under normal operation, one of the two head nodes is in *active* mode, whereas the other is in *passive* mode.

The difference between naming versus mode is illustrated by realizing that while a head node which is primary always remains primary, the mode that the node is in may change. Thus, the primary head node can be in passive mode when the secondary is in active mode. Similarly the primary head node may be in active mode while the secondary head node is in passive mode.

The difference between active and passive is that the active head takes the lead in cluster-related activity, while the passive follows it. Thus, for example, with MySQL transactions, CMDaemon carries them out with MySQL running on the active, while the passive trails the changes. This naturally means that the active corresponds to the master, and the passive to the slave, in the MySQL master-slave replication mode that MySQL is run as.

### 14.1.2 Monitoring The Active Head Node, Initiating Failover

In HA the passive head node continuously monitors the active head node. If the passive finds that the active is no longer operational, it will initiate a *failover sequence*. A failover sequence involves taking over resources, services and network addresses from the active head node. The goal is to continue providing services to compute nodes, so that jobs running on these nodes keep running.

### 14.1.3 Services In Bright Cluster Manager HA Setups

There are several services being offered by a head node to the cluster and its users.

**Services Running On Both Head Nodes**

One of the design features of the HA implementation in Bright Cluster Manager is that whenever possible, services are offered on both the active as well as the passive head node. This allows the capacity of both machines to be used for certain tasks (e.g. provisioning), but it also means that there are fewer services to move in the event of a failover sequence.

On a default HA setup, the following key services for cluster operations are always running on both head nodes:

- **CMDaemon:** providing certain functionality on both head nodes (e.g. provisioning)
- **DHCP:** load balanced setup
- **TFTP:** requests answered on demand, under xinetd
- **LDAP:** running in replication mode (the active head node LDAP database is pulled by the passive)
- **MySQL:** running in master-slave replication mode (the active head node MySQL database is pulled by the passive)
- **NTP**
• DNS

When an HA setup is created from a single head node setup, the above services are automatically reconfigured to run in the HA environment over two head nodes.

**Provisioning role runs on both head nodes** In addition, both head nodes also take up the *provisioning role*, which means that nodes can be provisioned from both head nodes. As the passive head node is then also provisioned from the active, and the active can switch between primary and secondary, it means both heads are also given a value for *provisioning interface* (section 5.4.7).

For a head node in a single-headed setup, there is no value set by default. For head nodes in an HA setup, the value of *provisioning interface* for each head node is automatically set up by default to the interface device name over which the image can be received when the head node is passive.

The implications of running a cluster with multiple provisioning nodes are described in detail in section 5.2. One important aspect described in that section is how to make provisioning nodes aware of image changes.

From the administrator’s point of view, achieving awareness of image changes for provisioning nodes in HA clusters is dealt with in the same way as for single-headed clusters. Thus, if using cmsh, the `updateprovisioners` command from within `softwareimage` mode is used, whereas if Bright View is used, then the clickpath *Provisioning → Provisioning requests → Update provisioning nodes* can be followed (section 5.2.4)

**Services That Migrate To The Active Node**

Although it is possible to configure any service to migrate from one head node to another in the event of a failover, in a typical HA setup only the following services migrate:

• NFS
• The User Portal
• Workload Management (e.g. SGE, Slurm)

**14.1.4 Failover Network Topology**

A two-head failover network layout is illustrated in figure 14.1.
In the illustration, the primary head1 is originally a head node before the failover design is implemented. It is originally set up as part of a Type 1 network (section 3.3.7 of the Installation Manual), with an internal interface eth0, and an external interface eth1.

When the secondary head is connected up to help form the failover system, several changes are made.

**HA: Network Interfaces**

Each head node in an HA setup typically has at least an external and an internal network interface, each configured with an IP address.

In addition, an HA setup uses two virtual IP interfaces, each of which has an associated virtual IP address: the external shared IP address and the internal shared IP address. These are shared between the head nodes, but only one head node can host the address and its interface at any time.

In a normal HA setup, a shared IP address has its interface hosted on the head node that is operating in active mode. On failover, the interface migrates and is hosted on the head node that then becomes active.

When head nodes are also being used as login nodes, users outside of the cluster are encouraged
to use the shared external IP address for connecting to the cluster. This ensures that they always reach whichever head node is active. Similarly, inside the cluster, nodes use the shared internal IP address wherever possible for referring to the head node. For example, nodes mount NFS filesystems on the shared internal IP interface so that the imported filesystems continue to be accessible in the event of a failover.

Shared interfaces are implemented as alias interfaces on the physical interfaces (e.g. eth0:0). They are activated when a head node becomes active, and deactivated when a head node becomes passive.

**HA: Dedicated Failover Network**

In addition to the normal internal and external network interfaces on both head nodes, the two head nodes are usually also connected using a direct dedicated network connection, eth2 in figure 14.1. This connection is used between the two head nodes to monitor their counterpart’s availability. It is called a heartbeat connection because the monitoring is usually done with a regular heartbeat-like signal between the nodes such as a ping, and if the signal is not detected, it suggests a head node is dead.

To set up a failover network, it is highly recommended to simply run a UTP cable directly from the NIC of one head node to the NIC of the other, because not using a switch means there is no disruption of the connection in the event of a switch reset.

**14.1.5 Shared Storage**

Almost any HA setup also involves some form of shared storage between two head nodes to preserve state after a failover sequence. For example, user home directories must always be available to the cluster in the event of a failover.

In the most common HA setup, the following two directories are shared:

- `/home`, the user home directories
- `/cm/shared`, the shared tree containing applications and libraries that are made available to the nodes

The shared filesystems are only available on the active head node. For this reason, it is generally recommended that users login via the shared IP address, rather than ever using the direct primary or secondary IP address. End-users logging into the passive head node by direct login may run into confusing behavior due to unmounted filesystems.

Although Bright Cluster Manager gives the administrator full flexibility on how shared storage is implemented between two head nodes, there are generally three types of storage used: NAS, DAS, and DRBD.

**NAS**

In a Network Attached Storage (NAS) setup, both head nodes mount a shared volume from an external network attached storage device. In the most common situation this would be an NFS server either inside or outside of the cluster.

Because imported mounts can typically not be re-exported (which is true at least for NFS), nodes typically mount filesystems directly from the NAS device.

**DAS**

In a Direct Attached Storage (DAS) setup, both head nodes share access to a block device that is usually accessed through a SCSI interface. This could be a disk-array that is connected to both head nodes, or it could be a block device that is exported by a corporate SAN infrastructure.

Although the block device is visible and can physically be accessed simultaneously on both head nodes, the filesystem that is used on the block device is typically not suited for simultaneous access. Simultaneous access to a filesystem from two head nodes must therefore be avoided because it generally leads to filesystem corruption. Only special purpose parallel filesystems such as GPFS and Lustre are capable of being accessed by two head nodes simultaneously.

© Bright Computing, Inc.
DRBD

The use of DRBD is not recommended for Bright Cluster Manager, and RHEL7 and CentOS7 no longer provide packages for it.

In a setup with DRBD (Distributed Replicated Block Device), both head nodes mirror a physical block device on each node device over a network interface. This results in a virtual shared DRBD block device. A DRBD block device is effectively a DAS block device simulated via a network. DRBD is a cost-effective solution for implementing shared storage in an HA setup. While a DRBD device itself is not configured by the cluster manager, a DRBD device that is already configured is recreated by the cluster manager. Recreating means that data on the DRBD device is wiped.

Custom Shared Storage With Mount And Unmount Scripts

The cluster management daemon on the two head nodes deals with shared storage through a mount script and an unmount script. When a head node is moving to active mode, it must acquire the shared filesystems. To accomplish this, the other head node first needs to relinquish any shared filesystems that may still be mounted. After this has been done, the head node that is moving to active mode invokes the mount script which has been configured during the HA setup procedure. When an active head node is requested to become passive (e.g. because the administrator wants to take it down for maintenance without disrupting jobs), the unmount script is invoked to release all shared filesystems.

By customizing the mount and unmount scripts, an administrator has full control over the form of shared storage that is used. Also an administrator can control which filesystems are shared.

Mount scripts paths can be set via cmsh or Bright View (section 14.4.6).

14.1.6 Guaranteeing One Active Head At All Times

Because of the risks involved in accessing a shared filesystem simultaneously from two head nodes, it is vital that only one head node is in active mode at any time. To guarantee that a head node that is about to switch to active mode will be the only head node in active mode, it must either receive confirmation from the other head node that it is in passive mode, or it must make sure that the other head node is powered off.

What Is A Split Brain?

When the passive head node determines that the active head node is no longer reachable, it must also take into consideration that there could be a communication disruption between the two head nodes. Because the “brains” of the cluster are communicatively “split” from each other, this is called a split brain situation.

Since the normal communication channel between the passive and active may not be working correctly, it is not possible to use only that channel to determine either an inactive head or a split brain with certainty. It can only be suspected.

Thus, on the one hand, it is possible that the head node has, for example, completely crashed, becoming totally inactive and thereby causing the lack of response. On the other hand, it is also possible that, for example, a switch between both head nodes is malfunctioning, and that the active head node is still up and running, looking after the cluster as usual, and that the head node in turn observes that the passive head node seems to have split away from the network.

Further supporting evidence from the dedicated failover network channel is therefore helpful. Some administrators find this supporting evidence an acceptable level of certainty, and configure the cluster to decide to automatically proceed with the failover sequence, while others may instead wish to examine the situation first before manually proceeding with the failover sequence. The implementation of automatic vs manual failover is described in section 14.1.7. In either implementation, fencing, described next, takes place until the formerly active node is powered off.

Going Into Fencing Mode

To deal with a suspected inactive head or split brain, a passive head node that notices that its active counterpart is no longer responding, first goes into fencing mode from that time onwards. While a node
is fencing, it will try to obtain proof via another method that its counterpart is indeed inactive.

Fencing, incidentally, does not refer to a thrust-and-parry imagery derived from fencing swordplay. Instead, it refers to the way all subsequent actions are tagged and effectively fenced-off as a backlog of actions to be carried out later. If the head nodes are able to communicate with each other before the passive decides that its counterpart is now inactive, then the fenced-off backlog is compared and synced until the head nodes are once again consistent.

**Ensuring That The Unresponsive Active Is Indeed Inactive**

There are two ways in which “proof” can be obtained that an unresponsive active is inactive:

1. By asking the administrator to manually confirm that the active head node is indeed powered off
2. By performing a power-off operation on the active head node, and then checking that the power is indeed off to the server. This is also referred to as a STONITH (Shoot The Other Node In The Head) procedure

It should be noted that just pulling out the power cable is not the same as a power-off operation (section 14.2.4).

Once a guarantee has been obtained that the active head node is powered off, the fencing head node (i.e. the previously passive head node) moves to active mode.

**Improving The Decision To Initiate A Failover With A Quorum Process**

While the preceding approach guarantees one active head, a problem remains.

In situations where the passive head node loses its connectivity to the active head node, but the active head node is communicating without a problem to the entire cluster, there is no reason to initiate a failover. It can even result in undesirable situations where the cluster is rendered unusable if, for example, a passive head node decides to power down an active head node just because the passive head node is unable to communicate with any of the outside world (except for the PDU feeding the active head node).

One technique used by Bright Cluster Manager to reduce the chances of a passive head node powering off an active head node unnecessarily is to have the passive head node carry out a quorum procedure. All nodes in the cluster are asked by the passive node to confirm that they also cannot communicate with the active head node. If more than half of the total number of nodes confirm that they are also unable to communicate with the active head node, then the passive head node initiates the STONITH procedure and moves to active mode.

**14.1.7 Automatic Vs Manual Failover**

Administrators have a choice between creating an HA setup with automatic or manual failover. In case of automatic failover, an active head node is powered off when it is no longer responding at all, and a failover sequence is initiated automatically.

In case of manual failover, the administrator is responsible for initiating the failover when the active head node is no longer responding. No automatic power off is done, so the administrator is asked to confirm that the previously active node is powered off.

For automatic failover to be possible, power control must be defined for both head nodes. If power control is defined for the head nodes, automatic failover is attempted by default. However, it is possible to disable automatic failover. In cmsh this is done by setting the disableautomaticfailover property, which is a part of the HA-related parameters (section 14.4.6):

```
[root@bright82 ~]# cmsh
[bright82]# partition failover base
[bright82->partition[base]->failover]# set disableautomaticfailover yes
[bright82->partition*[base*]->failover*]# commit
```

© Bright Computing, Inc.
With Bright View it is carried out via the clickpath Cluster→Partition[base]→Settings
→Failover→Disable automatic failover

If no power control has been defined, or if automatic failover has been disabled, or if the power
control mechanism is not working (for example due to inappropriate, broken or missing electronics or
hardware), then a failover sequence must always be initiated manually by the administrator.

In addition, if automatic failover is enabled, but the active head is still slightly responsive (the so-
called mostly dead state, described in section 14.4.2), then the failover sequence must also be initiated
manually by the administrator.

14.1.8 HA And Cloud Nodes
As far as the administrator is concerned, HA setup remains the same whether a Cluster Extension (Chap-
ter 3 of the Cloudbursting Manual) is configured or not, and whether a Cluster On Demand (Chapter 2 of
the Cloudbursting Manual) is configured or not. Behind the scenes, on failover, any networks associated
with the cloud requirements are taken care of by Bright Cluster Manager.

14.1.9 HA Using Virtual Head Nodes
Two physical servers are typically used for HA configurations. However, each head node can also be a
virtual machine (VM). The use case for this might be to gain experience with an HA configuration.

Failover Network Considerations With HA VMs
With physical head nodes in an HA configuration, the failover network, used for HA heartbeats, is
typically provided by running a network cable directly between the ethernet port on each machine. Not
having even a switch in between is a best practice. Since the head nodes are typically in the same, or
adjacent racks, setting this up is usually straightforward.

With VMs as head nodes in an HA configuration, however, setting up the failover network can be
more complex:

• The cluster administrator may need to consider if HA is truly improved by, for example, connect-
ing the failover network of the physical node to a switch, and then plumbing all that into all the
hypervisors as a vnic (in the case of VMware).

• Often a virtual switch would be used between the virtual head nodes, just because it is often easier.

• Not using a failover network is also an option, just as in the physical case.

• If one head node is on one hypervisor, and another is on a second hypervisor, then a standard
Bright Cluster Manager setup cannot have one head node carry out an automated failover STONITH
because it cannot contact the other hypervisor. So powering off the VM in the other hypervisor
would have to be done manually. An alternative to this, if automated failover is required, is to
create custom power scripts.

Bright Cluster Manager has no specific guidelines for the network configuration of HA with VMs. The
process of configuration is however essentially the same as for a physical node.

Size Considerations With HA VMs
A virtual head node in practice may be configured with fewer CPUs and memory than a physical head
node, just because such configurations are more common options in a VM setup than for a physical
setup, and cheaper to run. However, the storage requirement is the same as for a physical node. The
important requirement is that the head nodes should have sufficient resources for the cluster.

14.2 HA Setup Procedure Using cmha-setup
After installation (Chapter 3 of the Installation Manual) and license activation (Chapter 4 of the Installation
Manual) an administrator may wish to add a new head node, and convert Bright Cluster Manager from
managing an existing single-headed cluster to managing an HA cluster.
Is An HA-Enabled License Required?
To convert a single-headed cluster to an HA cluster, the existing cluster license should first be checked to see if it allows HA. The `verify-license` command run with the `info` option can reveal this in the MAC address field:

**Example**

```
verify-license info | grep ^MAC
```

HA-enabled clusters display two MAC addresses in the output. Single-headed clusters show only one.

If an HA license is not present, it should be obtained from the Bright Cluster Manager reseller, and then be activated and installed (Chapter 4 of the *Installation Manual*).

Existing User Certificates Become Invalid
Installing the new license means that any existing user certificates will lose their validity (page 76 of the *Installation Manual*) on Bright View session logout. This means:

- If LDAP is managed by Bright Cluster Manager, then on logout, new user certificates are generated, and a new Bright View login session picks up the new certificates automatically.
- For LDAPs other than that of Bright Cluster Manager, the user certificates need to be regenerated.

It is therefore generally good practice to have an HA-enabled license in place before creating user certificates and profiles if there is an intention of moving from a single-headed to an HA-enabled cluster later on.

The cmha-setup Utility For Configuring HA
The `cmha-setup` utility is a special tool that guides the administrator in building an HA setup from a single head cluster. It is not part of the cluster manager itself, but is a cluster manager tool that interacts with the cluster management environment by using `cmsh` to create an HA setup. Although it is in theory also possible to create an HA setup manually, using either Bright View or `cmsh` along with additional steps, this is not supported, and should not be attempted as it is error-prone.

A basic HA setup is created in three stages:

1. **Preparation** (section 14.2.1): the configuration parameters are set for the shared interface and for the secondary head node that is about to be installed.

2. **Cloning** (section 14.2.2): the secondary head node is installed by cloning it from the primary head node.

3. **Shared Storage Setup** (section 14.2.3): the method for shared storage is chosen and set up.

An optional extra stage is:

4. **Automated Failover Setup** (section 14.2.4): Power control to allow automated failover is set up.

14.2.1 Preparation
The following steps prepare the primary head node for the cloning of the secondary. The preparation is done only on the primary, so that the presence of the secondary is not actually needed during this stage.

0. It is recommended that all nodes except for the primary head node are powered off, in order to simplify matters. The nodes should in any case be power cycled or powered back on after the basic HA setup stages (sections 14.2.1-14.2.3, and possibly section 14.2.4) are complete.

1. If bonding (section 3.5) is to be used on the head node used in an HA setup, then it is recommended to configure and test out bonding properly before carrying out the HA setup.
2. To start the HA setup, the `cmha-setup` command is run from a root shell on the primary head node.

3. Setup is selected from the main menu (figure 14.2).

4. Configure is selected from the Setup menu.

5. A license check is done. Only if successful does the setup proceed further. If the cluster has no HA-enabled license, a new HA-enabled license must first be obtained from the Bright Cluster Manager reseller, and activated (section 4.3 of the Installation Manual).

6. The virtual shared internal alias interface name and virtual shared internal IP alias address are set.

7. The virtual shared external alias interface name and virtual shared external IP alias address are set. To use DHCP assignments on external interfaces, 0.0.0.0 can be used.

8. The host name of the passive is set.

9. Failover network parameters are set. The failover network physical interface should exist, but the interface need not be up. The network name, its base address, its netmask, and domain name are set. This is the network used for optional heartbeat monitoring.

10. Failover network interfaces have their name and IP address set for the active and passive nodes.

11. The primary head node may have other network interfaces (e.g. InfiniBand interfaces, a BMC interface, alias interface on the BMC network). These interfaces are also created on the secondary head node, but the IP address of the interfaces still need to be configured. For each such interface, when prompted, a unique IP address for the secondary head node is configured.

12. The network interfaces of the passive head node are reviewed and can be adjusted as required. DHCP assignments on external interfaces can be set by using an IP address of 0.0.0.0.

13. A summary screen displays the planned failover configuration. If alterations need to be made, they can be done via the next step.

14. The administrator is prompted to set the planned failover configuration. If it is not set, the main menu of `cmha-setup` is re-displayed.
15. If the option to set the planned failover configuration is chosen, then a password for the MySQL root user is requested. The procedure continues further after the password is entered.

16. Setup progress for the planned configuration is displayed (figure 14.3).

17. Instructions on what to run on the secondary to clone it from the primary are displayed (figure 14.4).

**Figure 14.3: cmha-setup Setup Progress For Planned Configuration**

**Figure 14.4: cmha-setup Instructions To Run On Secondary For Cloning**

### 14.2.2 Failover Cloning

There are actually two kinds of cloning possible. Both types use the /cm/cm-clone-install command described further on in this section. The two cloning types are:

- **Failover cloning**: A passive head node can be created from the active head node. This uses the --failover option to create a copy that is very similar to the active head node, but with changes to make it a passive head, ready for failover purposes.
The process described in this section PXE boots the passive from the active, thereby loading a special rescue image from the active that allows cloning from the active to the passive to take place. This section is therefore about failover cloning. How to carry out re-cloning is described in section 14.4.8.

After the preparation has been done by configuring parameters as outlined in section 14.2.1, the failover cloning of the head nodes is carried out. In the cloning instructions that follow, the active node refers to the primary node and the passive node refers to the secondary node. However this correlation is only true for when an HA setup is created for the first time, and it is not necessarily true if head nodes are replaced later on by cloning.

These cloning instructions may also be repeated later on if a passive head node ever needs to be replaced, for example, if the hardware is defective (section 14.4.8). In that case the active head node can be either the primary or secondary.

1. The passive head node is PXE booted off the internal cluster network, from the active head node. It is highly recommended that the active and passive head nodes have identical hardware configurations. The BIOS clock of the head nodes should match and be set to the local time. Typically, the BIOS of both head nodes is also configured so that a hard disk boot is attempted first, and a PXE boot is attempted after a hard disk boot failure, leading to the Cluster Manager PXE Environment menu of options. This menu has a 5s time-out.

2. In the Cluster Manager PXE Environment menu of the node that is to become a clone, before the 5s time-out, "Start Rescue Environment" is selected to boot the node into a Linux ramdisk environment.

3. Once the rescue environment has finished booting, a login as root is done. No password is required (figure 14.5).

4. The following command is executed (figure 14.6) on the node that is to become a failover clone:

   ```
   /cm/cm-clone-install --failover
   ```
14.2 HA Setup Procedure Using cmha-setup

When doing a re-clone as in section 14.4.8, instead of a failover clone, then it is the --clone option that is used instead of the --failover option.

```
ClusterManager login: root
Welcome to Linux 2.6.32-228.23.1.el6.x86_64.
No mail.
# /cm/cmha-clone-install --failover
Network interface to use [default: eth0]:
Please wait while authentication is being set up....
root@master’s password:
Please wait while installation begins...
Verifying license ......................... [ OK ]
Getting build config .................... [ OK ]
Getting disk layout .................... [ OK ]
The hard mode disk layout is saved in /cm/_masterdisksetup.xml
[v - view, e - edit, c - continue 1: c]
The contents of the following disks will be erased.
/dev/sda
Do you want to continue [yes/no]? yes
Getting mount points .................... [ OK ]
Partitioning hard drive ................ [ OK ]
Syncing hard drive .................... [ OK ]
Finalizing installation ................ [ OK ]
Do you want to reboot[y/n]? y
```

Figure 14.6: Cloning The Passive From The Active Via A Rescue Mode Session

5. When prompted to enter a network interface to use, the interface that was used to boot from the internal cluster network (e.g. eth0, eth1, ...) is entered. There is often uncertainty about what interface name corresponds to what physical port. This can be resolved by switching to another console and using “ethtool -p <interface>”, which makes the NIC corresponding to the interface blink.

6. If the provided network interface is correct, a root@master’s password prompt appears. The administrator should enter the root password.

7. An opportunity to view or edit the master disk layout is offered.

8. A confirmation that the contents of the specified disk are to be erased is asked for.

9. The cloning takes place. The “syncing” stage usually takes the most time. Cloning progress can also be viewed on the active by selecting the “Install Progress” option from the Setup menu. When viewing progress using this option, the display is automatically updated as changes occur.

10. After the cloning process has finished, a prompt at the console of the passive asks if a reboot is to be carried out. A “y” is typed in response to this. The passive node should be set to reboot off its hard drive. This may require an interrupt during reboot, to enter a change in the BIOS setting, if for example, the passive node is set to network boot first.

11. Continuing on now on the active head node, Finalize is selected from the Setup menu of cmha-setup.

12. The MySQL root password is requested. After entering the MySQL password, the progress of the Finalize procedure is displayed, and the cloning procedure continues.

13. The cloning procedure of cmha-setup pauses to offer the option to reboot the passive. The administrator should accept the reboot option. After reboot, the cloning procedure is complete. The administrator can then go to the main menu and quit from there or go on to configure “Shared Storage” (section 14.2.3) from there.

© Bright Computing, Inc.
A check can already be done at this stage on the failover status of the head nodes with the `cmha` command, run from either head node:

**Example**

```
[root@bright82 ~]# cmha status
Node Status: running in active master mode

Failover status:
bright82* -> master2
 failoverping [ OK ]
 mysql [ OK ]
  ping [ OK ]
 status [ OK ]
master2 -> bright82*
 failoverping [ OK ]
 mysql [ OK ]
  ping [ OK ]
 status [ OK ]
```

Here, the mysql, ping and status states indicate that HA setup completed successfully. The failoverping state uses the dedicated failover network route for its checks, and starts working as soon as the passive head node has been rebooted.

### 14.2.3 Shared Storage Setup

After cloning the head node (section 14.2.2), the last basic stage of creating an HA setup is setting up shared storage. The available shared storage forms are NAS and DAS. The use of DRBD is not recommended by Bright Cluster Manager, but is still available as a legacy form of shared storage at the time of writing (May 2018).

**NAS**

1. In the `cmha-setup` main menu, the “Shared Storage” option is selected.

2. NAS is selected.

3. The parts of the head node filesystem that are to be copied to the NAS filesystems are selected. By default, these are `/home` and `/cm/shared` as suggested in section 14.1.5. The point in the filesystem where the copying is done is the future mount path to where the NAS will share the shared filesystem.

   An already-configured export that is not shared is disabled in `/etc/exports` by `cmha-setup`. This is done to prevent the creation of stale NFS file handles during a failover. Sharing already-existing exports is therefore recommended. Storage can however be dealt with in a customized manner with mount and unmount scripts (page 584).

4. The NFS host name is configured. Also, for each head node filesystem that is to be copied to the NAS filesystem, there is an associated path on the NAS filesystem where the share is to be served from. These NFS volume paths are now configured.

5. If the configured NFS filesystems can be correctly mounted from the NAS server, the process of copying the local filesystems onto the NAS server begins.

**DAS**

A prerequisite to the DAS configuration steps that follow is that the partitions exist on the DAS device that is to be used for shared storage. These should be created manually if required, before running `cmha-setup`. © Bright Computing, Inc.
1. In the cmha-setup main menu, the “Shared Storage” option is selected.

2. DAS is selected.

3. The filesystems that are to be shared over the cluster are selected by the administrator. The filesystems that are shared are typically /cm/shared and /home, but this is not mandatory.

4. The filesystems that are to be shared are assigned DAS partitions by the administrator. For example, the administrator may specify these as /dev/sdc1 for /home and /dev/sdd3 for /cm/shared.

5. The administrator can choose to create a filesystem on the proposed DAS device.
   - Creating the filesystem on the device means any existing filesystems and files on the device are wiped during creation.
   - Otherwise, the existing filesystem and files on the DAS device remain.

6. A filesystem type is set from a choice of ext3, ext4, xfs.

7. A summary screen is presented showing the proposed changes.

8. After filesystems have been created, the current contents of the shared directories are copied onto the shared filesystems and the shared filesystems are mounted over the old non-shared filesystems.

9. The administrator should check that the partitions are visible from both head nodes using, for example, the fdisk -l command on the device. If the partitions on the DAS are created or modified, or appear only after the passive head is running due to a hardware-related reason after the passive head is powered on, then the kernel on the passive head may not have reread the partition table. A power cycle of the head nodes is recommended if the partitions are not seen properly.

**DRBD**

If a DRBD device is used that already has data on it, then it should be backed up by the administrator, and restored after the cmha-setup procedure is complete. This is because the device content is erased by re-creation of the device partitions that takes place during the procedure. The procedure is carried out as follows:

1. In the cmha-setup main menu, the “Shared Storage” option is selected.

2. DRBD is selected.

3. The parts of the filesystem that should be placed on DRBD filesystems are selected.

4. The host names of the primary and secondary head nodes and the physical disk partitions to use on both head nodes are entered.

5. That the contents of the listed partitions can be erased on both head nodes is confirmed. After DRBD based filesystems have been created, the current contents of the shared directories are copied onto the DRBD based filesystems and the DRBD based filesystems are mounted over the old non-shared filesystems.

6. Once the setup process has completed, “DRBD Status/Overview” is selected to verify the status of the DRBD block devices.

© Bright Computing, Inc.
14.2.4 Automated Failover And Relevant Testing

A power-off operation on the active head node server does not mean the same as just pulling out the power cable to the active head node. These actions typically have different effects, and should therefore not be confused with each other. During the power-off operation, the BMC remains up. However, in the case of pulling out the power cable, the BMC is typically turned off too. If the BMC is not reachable, then it means that verifying that the active head has been terminated is uncertain. This is because the data that CMDaemon can access implies a logical possibility that there is a network failure rather than a head node failure. CMDaemon therefore does not carry out an automatic failover if the power cable is pulled out.

For automatic failover to work, the two head nodes must be able to power off their counterpart. This is done by setting up power control (Chapter 4).

Testing If Power Control Is Working

The “device power status” command in cmsh can be used to verify that power control is functional:

Example

```
[master1]\% device power status -n mycluster1,mycluster2
apc03:21 ............ [ ON ] mycluster1
apc04:18 ............ [ ON ] mycluster2
```

Testing The BMC Interface Is Working

If a BMC (Baseboard Management Controller, section 3.7) such as IPMI or iLO is used for power control, it is possible that a head node is not able to reach its own BMC interface over the network. This is especially true when no dedicated BMC network port is used. In this case, cmsh -c "device power status" reports a failure for the active head node. This does not necessarily mean that the head nodes cannot reach the BMC interface of their counterpart. Pinging a BMC interface can be used to verify that the BMC interface of a head node is reachable from its counterpart.

Example

Verifying that the BMC interface of mycluster2 is reachable from mycluster1:

```
[root@mycluster1 ~]# ping -c 1 mycluster2.bmc.cluster
PING mycluster2.bmc.cluster (10.148.255.253) 56(84) bytes of data.
64 bytes from mycluster2.bmc.cluster (10.148.255.253): icmp_seq=1
ttl=64 time=0.033 ms
```

Verifying that the BMC interface of mycluster1 is reachable from mycluster2:

```
[root@mycluster2 ~]# ping -c 1 mycluster1.bmc.cluster
PING mycluster1.bmc.cluster (10.148.255.254) 56(84) bytes of data.
64 bytes from mycluster1.bmc.cluster (10.148.255.254): icmp_seq=1
ttl=64 time=0.028 ms
```

Testing Automated Failover Against A Simulated Crash

A normal (graceful) shutdown of an active head node, does not cause the passive to become active, because HA assumes a graceful failover means there is no intention to trigger a failover. To carry out testing of an HA setup with automated failover, it is therefore useful to simulate a kernel crash on one of the head nodes. The following command crashes a head node instantly:

```
echo c > /proc/sysrq-trigger
```

After the active head node freezes as a result of the crash, the passive head node powers off the machine that has frozen and switches to active mode. A hard crash like this can cause a database replication inconsistency when the crashed head node is brought back up and running again, this time passively,
Running cmha-setup Without Ncurses, Using An XML Specification

14.3.1 Why Run It Without Ncurses?

The text-based Ncurses GUI for cmha-setup is normally how administrators should set up a failover configuration.

The express mode of cmha-setup allows an administrator to skip the GUI. This is useful, for example, for scripting purposes and speeding deployment. A further convenience is that this mode uses a human-editable XML file to specify the network and storage definitions for failover.

Running cmha-setup without the GUI still requires some user intervention, such as entering the root password for MySQL. The intervention required is scriptable with, for example, Expect, and is minimized if relevant options are specified for cmha-setup from the -x options.

14.3.2 The Syntax Of cmha-setup Without Ncurses

The express mode (-x) options are displayed when “cmha-setup -h” is run. The syntax of the -x options is indicated by:

cmha-setup [ -x -c <configfile> [-s <type>] <-i|-f[-r]> [-p <mysqlrootpassword>] ]

The -x options are:

- `-c|--config <configfile>`: specifies the location of <configfile>, which is the failover configuration XML file for cmha-setup. The file stores the values to be used for setting up a failover head node. The recommended location is at /cm/local/apps/cluster-tools/ha/conf/failoverconf.xml.

- `-i|--initialize`: prepares a failover setup by setting values in the CMDaemon database to the values specified in the configuration file. This corresponds to section 14.2.1. The administrator is prompted for the MySQL root password unless the -p option is used. The -i option of the script then updates the interfaces in the database, and clones the head node in the CMDaemon database. After this option in the script is done, the administrator normally carries clones the passive node from the active, as described in steps 1 to 10 of section 14.2.2.

- `-f|--finalize`: After the passive node is cloned as described in steps 1 to 10 of section 14.2.2, the finalize option is run on the active node to run the non-GUI finalize procedure. This is the non-GUI version of steps 11 to 13 of section 14.2.2.

- `-r|--finalizereboot`: makes the passive reboot after the finalize step completes.

- `-s|--sharedstorage <type>`: specifies the shared storage <type> out of a choice of nas, das or drbd.

- `-p|--pass <mysqlrootpassword>`: specifies the MySQL root password. Leaving this out means the administrator is prompted to type in the password during a run of the cmha-setup script when using the -x options.
There is little attempt at validation with the express mode, and invalid entries can cause the command to hang.

### 14.3.3 Example cmha-setup Run Without Ncurses

#### Preparation And Initialization:

After shutting down all nodes except for the active head node, a configuration is prepared by the administrator in `/cm/local/apps/cluster-tools/ha/conf/failoverconf.xml`. The administrator then runs `cmha-setup` with the initialization option on the active:

```bash
[root@bright82 ~]# cd /cm/local/apps/cluster-tools/ha/conf
[root@bright82 conf]# cmha-setup -x -c failoverconf.xml -i
Please enter the mysql root password:
```

Initializing failover setup on master ...... [ OK ]
Updating shared internal interface ...... [ OK ]
Updating shared external interface ...... [ OK ]
Updating extra shared internal interfaces ...... [ OK ]
Updating failover network ...... [ OK ]
Updating primary master interfaces ...... [ OK ]
Cloning master node ...... [ OK ]
Updating secondary master interfaces ...... [ OK ]
Updating failover network interfaces ...... [ OK ]
Updating Failover Object ...... [ OK ]
```

The preceding corresponds to the steps in section 14.2.1.

#### PXE Booting And Cloning The Passive:

The passive head node is then booted up via PXE and cloned as described in steps 1 to 10 of section 14.2.2.

#### Finalizing On The Active And Rebooting The Passive:

Then, back on the active head node the administrator continues the session there, by running the finalization option with a reboot option:

```bash
[root@bright82 conf]# cmha-setup -x -c failoverconf.xml -f -r
Please enter the mysql root password:
```

Updating secondary master mac address ...... [ OK ]
Initializing failover setup on master2 ...... [ OK ]
Cloning database ...... [ OK ]
Update DB permissions ...... [ OK ]
Checking for dedicated failover network ...... [ OK ]
A reboot has been issued on master2

The preceding corresponds to steps 11 to 13 of section 14.2.2.

#### Adding Storage:

Continuing with the session on the active, setting up a shared storage could be done with:

```bash
[root@bright82 conf]# cmha-setup -x -c failoverconf.xml -s nas
```

The preceding corresponds to carrying out the NAS procedure of section 14.2.3.

### 14.4 Managing HA

Once an HA setup has been created, the tools in this section can be used to manage the HA aspects of the cluster.
14.4 Managing HA

14.4.1 Changing An Existing Failover Configuration
Changing an existing failover configuration is usually done most simply by running through the HA setup procedure of section 14.2 again, with one exception. The exception is that the existing failover configuration must be removed by using the “Undo Failover” menu option between steps 3 and 4 of the procedure described in section 14.2.1.

14.4.2 cmha Utility
A major command-line utility for interacting with the HA subsystem, for regular nodes as well as for head nodes, is cmha. It is part of the Bright Cluster Manager cluster-tools package. Its usage information is:

[root@mycluster1 ~]# cmha
Usage: cmha < status | makeactive [node] | dbreclone <host> | nodestatus [name] >

status Retrieve and print high availability status of head nodes.

nodestatus [groups] Retrieve and print high availability status of failover [groups] (comma separated list of group names. If no argument is given, then the status of all available failover groups is printed.

makeactive [node] Make the current head node the active head node. If [node] is specified, then make [node] the active node in the failover group that [node] is part of.

dbreclone <host> Clone MySQL database from this head node to <host> (hostname of failover head node).

Some of the information and functions of cmha can also be carried out via CMDaemon:

- For cmsh, the following commands can be run from within the base object in partition mode:
  - For the head node, the status and makeactive commands are run from within the failover submode.
  - For regular nodes the nodestatus and makeactive [node] commands are run from within the failovergroups submode.

The dbreclone option cannot be carried out in Bright View or cmsh because it requires stopping CMDaemon.

The cmha options status, makeactive, and dbreclone are looked at in greater detail next:

cmha status: Querying HA Status
Information on the failover status is displayed thus:

Example

[root@mycluster1 ~]# cmha status
Node Status: running in active master mode

Failover status:
mycluster1* -> mycluster2
  failoverping [ OK ]
  mysql [ OK ]

© Bright Computing, Inc.
ping [ OK ]
status [ OK ]
mycluster2 -> mycluster1
failoverping [ OK ]
mysql [ OK ]
ping [ OK ]
status [ OK ]

The * in the output indicates the head node which is currently active. The status output shows 4 aspects of the HA subsystem from the perspective of each head node:

<table>
<thead>
<tr>
<th>HA Status</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>failoverping</td>
<td>the other head node is reachable via the dedicated failover network. This failover ping uses the failover route instead of the internal net route. It uses ICMP ping.</td>
</tr>
<tr>
<td>mysql</td>
<td>MySQL replication status</td>
</tr>
<tr>
<td>ping</td>
<td>the other head node is reachable over the primary management network. It uses ICMP ping.</td>
</tr>
<tr>
<td>status</td>
<td>CMDaemon running on the other head node responds to REST calls</td>
</tr>
</tbody>
</table>

By default, Bright Cluster Manager prepares to carry out the failover sequence (the sequence that includes a STONITH) when all three of ping, failoverping and status are not OK on a head node. If these three are not OK, then the active node is all dead according to cmha. One way of initiating failover is thus by causing a system crash (section 14.2.4).

It can typically take about 30s for the cmha status command to output its findings in the case of a recently crashed head node.

cmha makeactive: **Initiate Failover**

If automatic failover is enabled (section 14.1.7), then the failover sequence attempts to complete automatically if power management is working properly, and the cmha status shows ping, failoverping and status as failed.

If automatic failover is disabled, then a manual failover operation must be executed to have a failover operation take place. A manual failover operation can be carried out with the “cmha makeactive” command:

**Example**

To initiate a failover manually:

```
[root@mycluster2 ~]# cmha makeactive
Proceeding will initiate a failover sequence which will make this node (mycluster2) the active master.

Are you sure ? [Y/N] 

y

Your session ended because: CMDaemon failover, no longer master
mycluster2 became active master, reconnecting your cmsh ...
```

On successful execution of the command, the former active head node simply continues to run as a passive head node.

The cmha makeactive command assumes both head nodes have no problems preventing the execution of the command. However, this may not be the case in an edge case, which requires more careful consideration before answering the Are you sure ? prompt.
cmha makeactive edge case—the mostly dead active:

- For a manual failover operation, if the execution of the `cmha makeactive` command has problems, then it can mean that there is a problem with the initially active head node being in a sluggish state. That is, neither fully functioning, nor all dead. The active head node is thus in a state that is still powered on, but what can be called mostly dead. Mostly dead means slightly alive (not all of ping, failoverping, and status are FAILED), while all dead means there is only one thing that can sensibly be done to make sure the cluster keeps running—that is, to make the old passive the new active.

Making an old passive the new active is only safe if the old active is guaranteed to not come back as an active head node. This guarantee is set by a STONITH (page 585) for the old active head node, and results in a former active that is now all dead. STONITH thus guarantees that head nodes are not in conflict about their active and passive states. STONITH can however still fail in achieving a clean shutdown when acting on a mostly dead active head node, which can result in unclean filesystem or database states.

Thus, the mostly dead active head node may still be in the middle of a transaction, so that shutting it down may cause filesystem or database corruption. Making the passive node also active then in this case carries risks such as mounting filesystems accidentally on both head nodes, or carrying out database transactions on both nodes. This can also result in filesystem and database corruption.

It is therefore left to the administrator to examine the situation for corruption risk. The decision is either to power off a mostly dead head node, i.e. STONITH to make sure it is all dead, or whether to wait for a recovery to take place. When carrying out a STONITH on the mostly dead active head node, the administrator must power it off before the passive becomes active for a manual failover to take place with minimal errors. The `cmha dbreclone` option may still be needed to restore a corrupted database after such a power off, after bringing the system back up.

- For an automated failover configuration, powering off the mostly dead active head node is not carried out automatically due to the risk of filesystem and database corruption. A mostly dead active node with automatic failover configuration therefore stays mostly dead either until it recovers, or until the administrator decides to do a STONITH manually to ensure it is all dead. Here, too, the `cmha dbreclone` option may still be needed to restore a corrupted database after such a power off, after bringing the system back up.

**cmha dbreclone: Cloning The CMDaemon Database**

The `dbreclone` option of `cmha` clones the CMDaemon state database from the head node on which `cmha` runs to the head node specified after the option. It is normally run in order to clone the database from the active head node to the passive—running it from the passive to the active can cause a loss of database entries. Running the `dbreclone` option can be used to retrieve the MySQL CMDaemon state database tables, if they are, for example, unsalvageably corrupted on the destination node, and the source node has a known good database state. Because it is resource intensive, it is best run when there are few or no users. It is typically only used by administrators after being instructed to do so by Bright support.

**Example**

```
[root@bright82 ~]# cmha status
Node Status: running in active master mode

Failover status:
bright82* -> head2
  failoverping [ OK ]
  mysql [ OK ]
  ping [ OK ]
```

© Bright Computing, Inc.
status [ OK ]
head2 -> bright82*
  failoverping [ OK ]
  mysql [FAILED] (11)
  ping [ OK ]
  status [ OK ]
[root@bright82 ~]# cmha dbreclone head2
Proceeding will cause the contents of the cmdaemon state database on he\
ad2 to be resynchronized from this node (i.e. bright82 -> head2)

Are you sure? [Y/N]
Y
Waiting for CMDaemon (3113) to terminate...
  [ OK ]
Waiting for CMDaemon (7967) to terminate...
  [ OK ]
cmdaemon.dump.8853.sql 100% 253KB 252.9KB/s 00:00
slurmacctdb.dump.8853.sql 100% 11KB 10.7KB/s 00:00
Waiting for CMDaemon to start...  [ OK ]
Waiting for CMDaemon to start...[ OK ]
[root@bright82 ~]# cmha status
Node Status: running in active master mode

Failover status:
  bright82* -> head2
    failoverping [ OK ]
    mysql [ OK ]
    ping [ OK ]
    status [ OK ]
  head2 -> bright82*
    failoverping [ OK ]
    mysql [ OK ]
    ping [ OK ]
    status [ OK ]

14.4.3 States
The state a head node is in can be determined in three different ways:

1 By looking at the message being displayed at login time.

   Example

   Node Status: running in active master mode

2 By executing cmha status.

   Example

   [root@mycluster ~]# cmha status
   Node Status: running in active master mode

   ...
3 By examining `/var/spool/cmd/state`.

There are a number of possible states that a head node can be in:

<table>
<thead>
<tr>
<th>State</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>INIT</td>
<td>Head node is initializing</td>
</tr>
<tr>
<td>FENCING</td>
<td>Head node is trying to determine whether it should try to become active</td>
</tr>
<tr>
<td>ACTIVE</td>
<td>Head node is in active mode</td>
</tr>
<tr>
<td>PASSIVE</td>
<td>Head node is in passive mode</td>
</tr>
<tr>
<td>BECOMEACTIVE</td>
<td>Head node is in the process of becoming active</td>
</tr>
<tr>
<td>BECOMEPASSIVE</td>
<td>Head node is in the process of becoming passive</td>
</tr>
<tr>
<td>UNABLETOBECOMEACTIVE</td>
<td>Head node tried to become active but failed</td>
</tr>
<tr>
<td>ERROR</td>
<td>Head node is in error state due to unknown problem</td>
</tr>
</tbody>
</table>

Especially when developing custom mount and unmount scripts, it is quite possible for a head node to go into the UNABLETOBECOMEACTIVE state. This generally means that the mount and/or unmount script are not working properly or are returning incorrect exit codes. To debug these situations, it is helpful to examine the output in `/var/log/cmdaemon`. The “cmha makeactive” shell command can be used to instruct a head node to become active again.

### 14.4.4 Failover Action Decisions

A table summarizing the scenarios that decide when a passive head should take over is helpful:
### High Availability

#### Event on active | Reaction on passive | Reason
--- | --- | ---
Reboot | Nothing | Event is usually an administrator action. To make the passive turn active, an administrator would run “cmha makeactive” on it.

Shutdown | Nothing | As above.

Unusually sluggish or freezing system by state pingable with ICMP packets | Nothing | 1. Active may still unfreeze. 2. Shared file systems may still be in use by the active. Concurrent use by the passive taking over therefore risks corruption. 3. Mostly dead head can be powered off by administrator after examining situation (section 14.4.2).

Become passive in response to “cmha makeactive” run on passive | Become active when former active becomes passive | As ordered by administrator

Active dies | Quorum called, may lead to passive becoming new active | Confirms if active head is dead according to other nodes too. If so, then a “power off” command is sent to it. If the command is successful, the passive head becomes the new active head.

### 14.4.5 Keeping Head Nodes In Sync

#### What Should Be Kept In Sync?
- It is a best practice to carry out a manual `updateprovisioners` command on the active head node, immediately after a regular node software image change has been made.

A successful run of the `updateprovisioners` command means that in the event of a failover, the formerly passive head node already has up-to-date regular node software images, which makes further administration simpler.

The background behind why it is done can be skipped, but it is as follows:

An image on the passive head node, which is a node with a provisioning role, is treated as an image on any other provisioning node. This means that it eventually synchronizes to a changed image on the active head node. By default the synchronization happens at midnight, which means images may remain out-of-date for up to 24 hours. The images being in an out-of-date state on the passive is normal, because the timeout period associated with being in an updated state is only 5 minutes by default.

Since the passive head is a provisioning node, it also means that an attempt to provision regular nodes from it with the changed image will not succeed if it happens too soon after the image change event on the active head. “Too soon” means within the autoupdate period defined by the parameter `provisioningnodeautoupdatetimout` (page 158).

If on the other hand the autoupdate timeout is exceeded, then by itself this does not lead to the image on the passive head node becoming synchronized with an image from the active head node. Such synchronization only takes place as part of regular housekeeping (at midnight by default).
Or it takes place if a regular node sends a provisioning request to the passive head node, which can take place during the reboot of the regular node.

This synchronization logic is followed to reduce the load on the head node. The only pitfall in this is the case when an administrator changes an image on the active head node, and then soon after that the passive head node becomes active as part of a failover, without the images having had enough time to synchronize. In that case the formerly passive node ends up with out-of-date software images. That is why it is a best practice to carry out a manual update-provisioners command on the active head immediately after a regular node software image change has been made.

- Changes controlled by CMDaemon are synchronized automatically between the CMDaemon databases to the required extent during failover to the active node.

  If the output of cmha status is not OK, then it typically means that the CMDaemon databases of the active head node and the passive head node are not synchronized. This situation may be resolved by waiting, typically for several minutes. If the status does not resolve on its own, then this indicates a more serious problem which should be investigated further.

- By default, software images are not stored on shared storage, and are synchronized between the head nodes by CMDaemon.

  However, if images are kept on shared storage, then, within the provisioning role (section 5.2.1), the image-related parameters such as allimages, localimages, and sharedimages, must be adjusted according to the configuration used.

- If filesystem changes are made on an active head node without using CMDaemon (cmsh or Bright View), and if the changes are outside the shared filesystem, then these changes should normally also be made by the administrator on the passive head node. For example:
  
  - RPM installations/updates (section 11.2)
  - Applications installed locally
  - Files (such as drivers or values) placed in the /cm/node-installer/ directory and referred to by initialize (section 5.4.5) and finalize scripts (section 5.4.11)
  - Any other configuration file changes outside of the shared filesystems

  The reason behind not syncing everything automatically is to guarantee that a change that breaks a head node is not accidentally propagated to the passive. This way there is always a running head node. Otherwise, if automated syncing is used, there is a risk of ending up with two broken head nodes at the same time.

  If the cluster is being built on bare metal, then a sensible way to minimize the amount of work to be done is to install a single head cluster first. All packages and applications should then be placed, updated and configured on that single head node until it is in a satisfactory state. Only then should HA be set up as described in section 14.2, where the cloning of data from the initial head node to the secondary is described. The result is then that the secondary node gets a well-prepared system with the effort to prepare it having only been carried out once.

**Avoiding Encounters With The Old Filesystems**

It should be noted that when the shared storage setup is made, the contents of the shared directories (at that time) are copied over from the local filesystem to the newly created shared filesystems. The shared filesystems are then mounted on the mountpoints on the active head node, effectively hiding the local contents.

  Since the shared filesystems are only mounted on the active machine, the old filesystem contents remain visible when a head node is operating in passive mode. Logging into the passive head node may thus confuse users and is therefore best avoided.

© Bright Computing, Inc.
Updating Services On The Head Nodes And Associated Syncing

The services running on the head nodes described in section 14.1.3 should also have their packages updated on both head nodes.

For the services that run simultaneously on the head nodes, such as CMDaemon, DHCP, LDAP, MySQL, NTP and DNS, their packages should be updated on both head nodes at about the same time. A suggested procedure is to stop the service on both nodes around the same time, update the service and ensure that it is restarted.

The provisioning node service is part of the CMDaemon package. The service updates images from the active head node to all provisioning nodes, including the passive head node, if the administrator runs the command to update provisioners. How to update provisioners is described in section 14.1.3.

For services that migrate across head nodes during failover, such as NFS and Workload Management, it is recommended (but not mandated) to carry out this procedure: the package on the passive node (called the secondary for the sake of this example) is updated to check for any broken package behavior. The secondary is then made active with cmha makeactive (section 14.4.2), which automatically migrates users cleanly off from being serviced by the active to the secondary. The package is then updated on the primary. If desired, the primary can then be made active again. The reason for recommending this procedure for services that migrate is that, in case the update has issues, the situation can be inspected somewhat better with this procedure.

### 14.4.6 High Availability Parameters

There are several HA-related parameters that can be tuned. Accessing these via Bright View is described in section 14.4.6. In cmsh the settings can be accessed in the failover submode of the base partition.

#### Example

```bash
[mycluster1] partition failover base
[mycluster1->partition[base]->failover] show
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dead time</td>
<td>10</td>
</tr>
<tr>
<td>Disable automatic failover</td>
<td>no</td>
</tr>
<tr>
<td>Failover network</td>
<td>failovernet</td>
</tr>
<tr>
<td>Init dead</td>
<td>30</td>
</tr>
<tr>
<td>Keep alive</td>
<td>1</td>
</tr>
<tr>
<td>Mount script</td>
<td></td>
</tr>
<tr>
<td>Postfailover script</td>
<td></td>
</tr>
<tr>
<td>Prefailover script</td>
<td></td>
</tr>
<tr>
<td>Quorum time</td>
<td>60</td>
</tr>
<tr>
<td>Revision</td>
<td></td>
</tr>
<tr>
<td>Secondary headnode</td>
<td></td>
</tr>
<tr>
<td>Unmount script</td>
<td></td>
</tr>
<tr>
<td>Warn time</td>
<td>5</td>
</tr>
</tbody>
</table>

### Dead time

When a passive head node determines that the active head node is not responding to any of the periodic checks for a period longer than the Dead time seconds, the active head node is considered dead and a quorum procedure starts. Depending on the outcome of the quorum, a failover sequence may be initiated.

#### Disable automatic failover

Setting this to yes disables automated failover. Section 14.1.7 covers this further.

#### Failover network

The Failover network setting determines which network is used as a dedicated network for the failoverping heartbeat check. The heartbeat connection is normally a direct cable from a NIC on one
head node to a NIC on the other head node. The network can be selected via tab-completion suggestions. By default, without a dedicated failover network, the possibilities are nothing, externalnet and internalnet.

Init dead
When head nodes are booted simultaneously, the standard Dead time might be too strict if one head node requires a bit more time for booting than the other. For this reason, when a head node boots (or more exactly, when the cluster management daemon is starting), a time of Init dead seconds is used rather than the Dead time to determine whether the other node is alive.

Keep alive
The Keep alive value is the time interval, in seconds, over which the passive head node carries out a check that the active head node is still up. If a dedicated failover network is used, 3 separate heartbeat checks are carried out to determine if a head node is reachable.

Mount script
The script pointed to by the Mount script setting is responsible for bringing up and mounting the shared filesystems.

Postfailover script
The script pointed to by the Postfailover script setting is run by cmdaemon on both head nodes. The script first runs on the head that is now passive, then on the head that is now active. It runs as soon as the former passive has become active. It is typically used by scripts mounting an NFS shared storage so that no more than one head node exports a filesystem to NFS clients at a time.

Prefailover script
The script pointed to by the Prefailover script setting is run by cmdaemon on both head nodes. The script first runs on the (still) active head, then on the (still) passive head. It runs as soon as the decision for the passive to become active has been made, but before the changes are implemented. It is typically used by scripts unmounting an NFS shared storage so that no more than one head node exports a filesystem to NFS clients at a time. When unmounting shared storage, it is very important to ensure that a non-zero exit code is returned if unmounting has problems, or the storage may become mounted twice during the Postfailover script stage, resulting in data corruption.

Quorum time
When a node is asked what head nodes it is able to reach over the network, the node has Quorum time seconds to respond. If a node does not respond to a call for quorum within that time, it is no longer considered for the results of the quorum check.

Secondary headnode
The Secondary headnode setting is used to define the secondary head node to the cluster.

Unmount script
The script pointed to by the Unmount script setting is responsible for bringing down and unmounting the shared filesystems.

Warn time
When a passive head node determines that the active head node is not responding to any of the periodic checks for a period longer than Warn time seconds, a warning is logged that the active head node might become unreachable soon.
14.4.7 Viewing Failover Via Bright View

Accessing cmsh HA Parameters (partition failover base) Via Bright View

The Bright View equivalents of the cmsh HA parameters in section 14.4.6 are accessed from the clickpath Cluster→Partition[base]→Settings →Failover

14.4.8 Re-cloning A Head Node

Some time after an HA setup has gone into production, it may become necessary to re-install one of the head nodes, for example if one of the head nodes were replaced due to hardware failure.

To re-clone a head node from an existing active head node, the head node hardware that is going to become the clone can be PXE-booted into the rescue environment, as described in section 14.2.2. Instead of running the cm-clone-install --failover command as in that section, the following command can be run:

```
[root@bright82 ~]# /cm/cm-clone-install --clone --hostname=<new host name>
```

The new host name can be the same as the original, because the clone is not run at the same time as the original anyway. The clone should not be run after cloning on the same network segment as the original, in order to prevent IP address conflicts.

If the clone is merely intended as a backup, then the clone hardware does not have to match the head node. For a backup, typically the most important requirement is then that a clone drive should not run out of space—that is, its drive should be as large as, or larger than the matching drive of the head node.

If the clone is to be put to work as a head node, then, if the MAC address of one of the head nodes has changed, it is typically necessary to request that the product key is unlocked, so that a new license can be obtained (section 4.3 of the Installation Manual).

Exclude Lists And Cloning

Some files are normally excluded from being copied across from the head node to the clone, because syncing them is not appropriate.

The following exclude files are read from inside the directory /cm/ on the clone node when the cm-clone-install command is run (step 4 in section 14.2.2).

- `excludelistnormal`: used to exclude files to help generate a clone of the other head node. It is read when running the cm-clone-install command without the --failover option.
- `excludelistfailover`: used to exclude files to help generate a passive head node from an active head node. It is read when running the cm-clone-install --failover command.

In a default cluster, there is no need to alter these exclude files. However some custom head node configurations may require appending a path to the list.

The cloning that is carried out is logged in /var/log/clone-install-log.

Exclude Lists In Perspective

The exclude lists `excludelistfailover` and `excludelistnormal` described in the preceding paragraphs should not be confused with the exclude lists of section 5.6.1. The exclude lists of section 5.6.1:

- `excludelistupdate`
- `excludelistfullinstall`
- `excludelistsyncinstall`
- `excludelistgrabnew`
- `excludelistgrab`
14.5 HA For Regular Nodes

HA for regular nodes is available from Bright Cluster Manager version 7.0 onwards.

14.5.1 Why Have HA On Regular Nodes?
HA for regular nodes can be used to add services to the cluster and make them HA. Migrating the default existing HA services that run on the head node is not recommended, since these are optimized and integrated to work well as is. Instead, good candidates for this feature are other, extra, services that can run, or are already running, on regular nodes, but which benefit from the extra advantage of HA.

14.5.2 Comparing Head And Regular Node HA
Many of the features of HA for regular nodes are as for head nodes. These include:

### Head And Regular Node HA: Some Features In Common

- Power control is needed for all HA nodes, in order to carry out automatic failover (section 14.1.7).
- Warn time and dead time parameters can be set (section 14.4.6).
- Mount and unmount scripts (page 584).
- Pre- and post- failover scripts (section 14.4.6).
- Disabling or enabling automatic failover (section 14.1.7).

A virtual shared IP address that is presented as the virtual node that is always up (section 14.1.4).

Some differences between head and regular node HA are:

© Bright Computing, Inc.
### Head And Regular Node HA: Some Features That Differ

<table>
<thead>
<tr>
<th>Head Node HA</th>
<th>Regular Node HA</th>
</tr>
</thead>
<tbody>
<tr>
<td>Only one passive node.</td>
<td>Multiple passive nodes defined by failover groups.</td>
</tr>
<tr>
<td>Can use the optional failover network and failoverping heartbeat.</td>
<td>No failover network. Heartbeat checks done via regular node network.</td>
</tr>
<tr>
<td>Configured within failover submode of partition mode (section 14.4.6).</td>
<td>Configured within failovergroups submode of partition mode (section 14.5.3).</td>
</tr>
<tr>
<td>Installed with cmha-setup (section 14.2).</td>
<td>Installed by administrator using a procedure similar to section 14.5.3.</td>
</tr>
<tr>
<td>A quorum procedure (section 14.1.6). If more than half the nodes can only connect to the passive, then the passive powers off the active and becomes the new active.</td>
<td>Active head node does checks. If active regular node is apparently dead, it is powered off (STONITH). Another regular node is then made active.</td>
</tr>
</tbody>
</table>

### Failover Groups

Regular nodes use *failover groups* to identify nodes that are grouped for HA. Two or more nodes are needed for a failover group to function. During normal operation, one member of the failover group is active, while the rest are passive. A group typically provides a particular service.

#### 14.5.3 Setting Up A Regular Node HA Service

In *cmsh* a regular node HA service, CUPS in this example, can be set up as follows:

**Making The Failover Group**

A failover group must first be made, if it does not already exist:

**Example**

```
[bright82->partition[base]->failovergroups]% status
No active failover groups
[bright82->partition[base]->failovergroups]% add cupsgroup
[bright82->partition*[base*]->failovergroups*[cupsgroup*]]% list
Name (key) Nodes
------------------------ ------------------------
cupsgroup
```

By default, CUPS is provided in the standard image, in a stopped state. In Bright View a failover group can be added via the clickpath Cluster → Partition[base] → Settings → Failover groups → Add

**Adding Nodes To The Failover Group**

Regular nodes can then be added to a failover group. On adding, Bright Cluster Manager ensures that one of the nodes in the failover group becomes designated as the active one in the group (some text elided):

**Example**
14.5 HA For Regular Nodes

[bright82->...[cupsgroup*]]% set nodes node001..node002
[bright82->...[cupsgroup*]]% commit
[bright82->...[cupsgroup]]%
...Failover group cupsgroup, make node001 become active
...Failover group cupsgroup, failover complete. node001 became active
[bright82->partition[base]->failovergroups[cupsgroup]]%

Setting Up A Server For The Failover Group

The CUPS server needs to be configured to run as a service on all the failover group nodes. The usual way to configure the service is to set it to run only if the node is active, and to be in a stopped state if the node is passive:

Example

[bright82->partition[base]->failovergroups[cupsgroup]]% device
[bright82->device]% foreach -n node001..node002 (services; add cups; \ set runif active; set autostart yes; set monitored yes)
[bright82->device]% commit
Successfully committed 2 Devices
[bright82->device]%
Mon Apr 7 08:45:54 2014 [notice] node001: Service cups was started

The runif options are described in section 3.11.1.

Setting And Viewing Parameters And Status In The Failover Group

Knowing which node is active: The status command shows a summary of the various failover groups in the failovergroups submode, including which node in each group is currently the active one:

Example

[bright82->partition[base]->failovergroups[cupsgroup]]% status

<table>
<thead>
<tr>
<th>Name</th>
<th>State</th>
<th>Active</th>
<th>Nodes</th>
</tr>
</thead>
<tbody>
<tr>
<td>cupsgroup</td>
<td>ok</td>
<td>node001</td>
<td>node001,node002 [ UP ]</td>
</tr>
</tbody>
</table>

Making a node active: To set a particular node to be active, the makeactive command can be used from within the failover group:

Example

[bright82->partition[base]->failovergroups[cupsgroup]]% use cupsgroup
[bright82->...[cupsgroup]]% makeactive node002
node002 becoming active ...
[bright82->partition[base]->failovergroups[cupsgroup]]%
... Failover group cupsgroup, make node002 become active
...node001: Service cups was stopped
...node002: Service cups was started
...Failover group cupsgroup, failover complete. node002 became active

An alternative is to simply use the cmha utility (section 14.4.2):

Example

[root@bright82 ~]# cmha makeactive node002
Parameters for failover groups: Some useful regular node HA parameters for the failover group object, cupsgroup in this case, can be seen with the show command:

Example

```
[bright82->partition[base]->failovergroups]% show cupsgroup
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Automatic failover after graceful shutdown</td>
<td>no</td>
</tr>
<tr>
<td>Dead time</td>
<td>10</td>
</tr>
<tr>
<td>Disable automatic failover</td>
<td>no</td>
</tr>
<tr>
<td>Mount script</td>
<td>cupsgroup</td>
</tr>
<tr>
<td>Nodes</td>
<td>node001,node002</td>
</tr>
<tr>
<td>Postfailover script</td>
<td></td>
</tr>
<tr>
<td>Prefailover script</td>
<td></td>
</tr>
<tr>
<td>Revision</td>
<td></td>
</tr>
<tr>
<td>Unmount script</td>
<td></td>
</tr>
<tr>
<td>Warn time</td>
<td>5</td>
</tr>
</tbody>
</table>

Setting Up The Virtual Interface To Make The Server An HA Service

The administrator then assigns each node in the failover group the same alias interface name and IP address dotted quad on its physical interface. The alias interface for each node should be assigned to start up if the node becomes active.

Example

```
[bright82->device]% foreach -n node001..node002 (interfaces; add alias \bootif:0 ; set ip 10.141.240.1; set startif active; set network internalnet)
[bright82->device*]% commit
Successfully committed 2 Devices
```

```
[bright82->device]% foreach -n node001..node002 (interfaces; list)
```

<table>
<thead>
<tr>
<th>Type</th>
<th>Network device name</th>
<th>IP</th>
<th>Network</th>
</tr>
</thead>
<tbody>
<tr>
<td>alias</td>
<td>BOOTIF:0</td>
<td>10.141.240.1</td>
<td>internalnet</td>
</tr>
<tr>
<td>physical</td>
<td>BOOTIF [prov]</td>
<td>10.141.0.1</td>
<td>internalnet</td>
</tr>
</tbody>
</table>

Optionally, each alias node interface can conveniently be assigned a common arbitrary additional host name, perhaps associated with the server, which is CUPS. This does not result in duplicate names here because only one alias interface is active at a time. Setting different additional hostnames for the alias interface to be associated with a unique virtual IP address is not recommended.

Example

```
[bright82->...interfaces*[BOOTIF:0*]]% set additionalhostnames cups
[bright82->...interfaces*[BOOTIF:0*]]% commit
```

The preceding can also simply be included as part of the set commands in the foreach statement earlier when the interface was created.

The nodes in the failover group should then be rebooted.

Only the virtual IP address should be used to access the service when using it as a service. Other IP addresses may be used to access the nodes that are in the failover group for other purposes, such as monitoring or direct access.
Service Configuration Adjustments

A service typically needs to have some modifications in its configuration done to serve the needs of the cluster.

CUPS uses port 631 for its service and by default it is only accessible to the local host. Its default configuration is modified by changing some directives within the cupsd.conf file. For example, some of the lines in the default file may be:

```
# Only listen for connections from the local machine.
Listen localhost:631
...
# Show shared printers on the local network.
... BrowseLocalProtocols
...
<Location />
  # Restrict access to the server...
  Order allow,deny
</Location>
...
```

Corresponding lines in a modified cupsd.conf file that accepts printing from hosts on the internal network could be modified and end up looking like:

```
# Allow remote access
Port 631
...
# Enable printer sharing and shared printers.
... BrowseAddress @LOCAL
BrowseLocalProtocols CUPS dnssd
... <Location />
  # Allow shared printing...
  Order allow,deny
  Allow from 10.141.0.0/16
</Location>
...
```

The operating system that ends up on the failover group nodes should have the relevant service modifications running on those nodes after these nodes are up. In general, the required service modifications could be done:

- with an initialize or finalize script, as suggested for minor modifications in section 3.15.4
- by saving and using a new image with the modifications, as suggested for greater modifications in section 3.15.2, page 128.

Testing Regular Node HA

To test that the regular node HA works, the active node can have a simulated crash carried out on it like in section 14.2.4.

Example
ssh node001
echo c > /proc/sysrq-trigger
~.

A passive node then takes over.

### 14.5.4 The Sequence Of Events When Making Another HA Regular Node Active

The active head node tries to initiate the actions in the following sequence, after the `makeactive` command is run (page 609):

<table>
<thead>
<tr>
<th>Sequence Of Events In Making Another HA Regular Node Active</th>
</tr>
</thead>
<tbody>
<tr>
<td>All</td>
</tr>
<tr>
<td>Active</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>Passive</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>All</td>
</tr>
</tbody>
</table>

The actions are logged by CMDaemon.

The following conditions hold for the sequence of actions:

- The remaining actions are skipped if the active `umount` script fails.
- The sequence of events on the initial active node is aborted if a STONITH instruction powers it off.
- The actions for all nodes is done for that particular failover group, for all nodes in that group.

### 14.6 HA And Workload Manager Jobs

Workload manager jobs continue to run through a failover handover if conditions allow it.

The 3 conditions that must be satisfied are:

1. The workload manager setup must have been carried out
   
   (a) during initial installation
   
   or
   
   (b) during a run of `wlm-setup`
   
2. The HA storage setup must support the possibility of job continuity for that workload manager. This support is possible for the workload manager and HA storage pairings indicated by the following table:
Table 14.6: HA Support For Jobs Vs Shared Filesystems

<table>
<thead>
<tr>
<th>WLM</th>
<th>DRBD</th>
<th>DAS</th>
<th>NAS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slurm</td>
<td>Y</td>
<td>Y</td>
<td>Y</td>
</tr>
<tr>
<td>OGS</td>
<td>Y</td>
<td>Y</td>
<td>Y</td>
</tr>
<tr>
<td>UGE</td>
<td>Y</td>
<td>Y</td>
<td>Y</td>
</tr>
<tr>
<td>PBSPro</td>
<td>N</td>
<td>N</td>
<td>Y</td>
</tr>
<tr>
<td>LSF</td>
<td>N</td>
<td>N</td>
<td>Y</td>
</tr>
</tbody>
</table>

As the table shows, PBS Pro, and LSF are not able to support HA for jobs on DRBD or DAS filesystems. This is because they require daemons to run on the passive and active nodes at the same time, in order to provide access to `/cm/shared` at all times. During failover the daemons normally cannot run on the node that has failed, which means that the DAS and DRBD storage types cannot provide access to `/cm/shared` during this time. Job continuity cannot therefore be supported for these workload manager and storage type combinations.

3. Jobs must also not fail due to the shared filesystem being inaccessible during the short period that it is unavailable during failover. This usually depends on the code in the job itself, rather than the workload manager, since workload manager clients by default have timeouts longer than the dead time during failover.

Already-submitted jobs that are not yet running continue as they are, and run when the resources become available after the failover handover is complete, unless they fail as part of the Bright Cluster Manager pre-job health check configuration.
15 Dell BIOS Management

15.1 Introduction

Dell BIOS management in Bright Cluster Manager means that for nodes that run on Dell hardware, the BIOS settings and BIOS firmware updates can be managed via the standard Bright front end utilities to CMDaemon, Bright View and cmsh.

In turn, CMDaemon configures the BIOS settings and applies firmware updates to each node via a standard Dell utility called racadm. The racadm utility is part of the Dell OpenManage software stack.

The Dell hardware supported is listed in the following table:

<table>
<thead>
<tr>
<th>Models</th>
</tr>
</thead>
<tbody>
<tr>
<td>14G</td>
</tr>
<tr>
<td>R430, R630, R730, R730XD, R930</td>
</tr>
<tr>
<td>FC430, FC630, FC830</td>
</tr>
<tr>
<td>M630, M830</td>
</tr>
<tr>
<td>C6320</td>
</tr>
</tbody>
</table>

Bright Cluster Manager supports iDRAC Express and iDRAC Enterprise for these models too. This chapter describes the Dell BIOS management integration features, with:

- An introduction to the integration features (section 15.1)
- The prerequisites for the integration to work (section 15.2)
- A description of settings management in Bright View and cmsh (section 15.3)
- A troubleshooting FAQ (section 15.4)

15.2 Prerequisites For BIOS Management

- The utility racadm must be present on the Bright Cluster Manager head node. The utility is installed on the head node if Dell is selected as the node hardware manufacturer during Bright Cluster Manager installation (section 3.3.6 of the Installation Manual).

- IPMI must be working on all of the servers. This means that it should be possible to communicate out-of-band from the head node to all of the compute nodes, via the IPMI IP address.

- iDRAC management privileges must be enabled for all the nodes for the Bright BMC user (username: bright, userid: 4). This operation has to performed manually. This privilege cannot be enabled during node provisioning by Bright during the BMC configuration step.
15.3 BIOS settings

15.3.1 Initializing The BIOS Settings Via cmsh
Initially, by default, there are no values set for the BIOS settings in the nodes or categories. An administrator therefore typically imports the values for the nodes from the cluster, in order for Bright Cluster Manager to have an initial configuration. An example of how each node in Bright Cluster Manager can have its BIOS values defined is:

Example

[bright82->device] % foreach -n node002..node008 (dellsettings; importlive; commit; )

Similarly, a category can have BIOS settings defined by taking a single node with suitable settings, and running the importlive command with that node being used as the settings for that category.

Example

[bright82->category[default]] % dellsettings; importlive node009; commit

As usual with CMDaemon values, settings at the node level override settings at the category level. To clear settings at the node level for each node within a category default, the clearsettings command can be used:

Example

[bright82->device] % foreach -c default (dellsettings; clearsettings; commit; )

The importlive operation can only be performed from within cmsh at the time of writing (December 2017). The operation will also become available in Bright View updates in Bright Cluster Manager 8.2 later on.

15.3.2 Managing The BIOS Settings Via Bright View
The Dell BIOS settings can be accessed via Bright View, using settings within the node or within the category, using the clickpaths:
Devices → Nodes → Edit → Settings → Dell Settings
or:
Grouping → Node Categories → Settings → Dell Settings

If CMDaemon is able to communicate with the racadm utility the Dell Settings can then be configured (figure 15.1).
Some extra Dell-related fields can also be seen in the SYS INFO COLLECTOR subpane for a node, under the clickpath:
Devices → Nodes → Edit → Settings → System Overview
The motherboard manufacturer, name, and system manufacturer are among the values collected in this subpane.
15.3 BIOS settings

**The Dell Settings Subwindow**

![Dell Settings Subwindow](image)

**Figure 15.1:** Selection of a node node001, then opening the Dell Settings subwindow

The BIOS Settings subwindow gives access to some basic BIOS properties, including:

- (figure 15.1) boot order, whether to apply the configuration changes immediately or on the next boot, and serial communication settings
- (figure 15.2) a devicefirmware subwindow, where the BIOS firmware can be set.
- (figure 15.2) a nics subwindow, where the NIC properties can be viewed and set.
- (figure 15.3) performance and power profiles, processor frequency and memory refresh rates.

![Devicefirmware and NICS Subwindow](image)

**Figure 15.2:** devicefirmware and nics subwindow access from the Dell Settings subwindow

© Bright Computing, Inc.
The **devicefirmware** subwindow

The **devicefirmware** subwindow (figure 15.4) allows a path to be set to the location on the head node where the firmware has been placed by the administrator. This path is then used when a firmware update is carried out with the **devicefirmware** subwindow for nodes or node categories.

HTTP, FTP, or TFTP access can also be specified using IPV4 addresses. These options assume that a firmware Catalog file exist at the HTTP, FTP or TFTP servers that host the firmware files. A firmware update is carried out with the **racadm** update command. The **racadm** software only updates particular kinds of firmware. The **racadm** documentation should be refered to for supported firmware file formats.

**Example**
15.3 BIOS settings

/cm/shared/firmwares/BIOS_YKH49_WN64_1.3.6.EXE

tftp://10.2.4.3

ftp://10.2.4.3

http://10.2.4.3

For FTP access, a username and password should be set.

15.3.3 Applying Dell Settings And Firmware Updates Via Bright View

To save modifications to the Dell settings, the usual way of saving the properties is followed. In this case it is the Save button in the window that opens up the Dell Settings subwindow, that is used to save the properties.

15.3.4 Managing The BIOS Settings Via cmsh

In cmsh, the Dell BIOS settings are available under the dellsettings submode. The dellsettings submode is available under a node or category object. If no Dell settings are yet configured, then a new local object is created automatically. The new object is saved using commit.

BIOS: Listing Properties

The Dell BIOS properties can be viewed with the show command run within the submode:

Example

[bright82->category[default]->dellsettings]% show

Parameter Value
--------------------------------- ------------------------------------------------
Revision
applyImmediately no
biosBootOrder
collaborativeCpuPerfCtrl no
devicefirmware <submode>
energyEfficientTurbo no
energyPerformanceBias MaxPower
failsafeBaudRate 115200
ioNonPostedPrefetch yes
l3Cache no
logicalProcessor no
memFrequency MaxPerf
memPatrolScrub Standard
memRefreshRate 1x
mmioAbove4Gb no
modelName
monitorMWait no
nics <0 in submode>
nodeInterleaving no
numberOfCores
proc1TurboCoreNum All
proc2TurboCoreNum All
procC1E no
procCStates no
procPwrPerf MaxPerf
procTurboMode no
redirectionAfterBoot no

© Bright Computing, Inc.
remoteTerminal: Vt100Vt220
serialCommunication: Off
serialPort: Serial1Com1Serial2Com2
serviceTag
snoopMode: HomeSnoop
sysprofile: Custom
turboBoost: no
uncoreFrequency: DynamicUFS

**BIOS: Setting Properties**
Tab-completion suggestions for the set show the possible properties that can be modified.

**Example**

```
[bright82->category[default]->dellsettings]% set <TAB><TAB>
applyimmediately collaborativecpuperfctrl monitormwait ...
...
```

Tab-completion suggestions for a particular property, such as monitormwait, show the possible values that can be taken by a property with a selection of options. A property can then be set and committed:

**Example**

```
[bright82->category[default]->dellsettings]% set monitormwait <TAB><TAB>
no yes
[bright82->category[default]->dellsettings]% set monitormwait yes
[bright82->category[default]->dellsettings]% commit
```

**BIOS: Importing Properties With importlive**
The importlive command imports the Dell settings for a category from an existing node configuration.

**Example**

```
[bright82->category*[default*]->dellsettings]% help importlive
importlive - Imports current live dell settings
Usage:
   [-f] importlive node001
Options:
   -f, --forcerefresh
       Force cmdaemon to refresh its cached live DellSettings copy, before importing

Examples:
   importlive Imports current live dell settings

[bright82->category*[default*]->dellsettings]% importlive node001
[bright82->category*[default*]->dellsettings]% importlive node001
[bright82->category[default]->dellsettings]% commit
```

**Firmware Updates**
Firmware update configurations can be listed and set within the firmware submode:

**Example**
Applying Settings For Dell With `applydellsettings`

The `applydellsettings` command of cmsh is used to apply BIOS or firmware configuration changes to a node. The command is available from within device mode.

When `applydellsettings` is run, a background task is immediately automatically started to apply the settings. The progress can be monitored with the task submode, and the list of current background tasks can be viewed using the task list command.

Example

```
[bright82->device[node001]]% applydellsettings
[bright82->device[node001]]% task list

Details for each task can be shown with the show command, run from within the task submode.

Example

```
[bright82->task]% show 1

... If applying a Dell settings task has failed, then the Log property of the task should show a detailed error message explaining why.

Example

```
[bright82->task]% show 1

... Log: ERROR: Unable to connect to RAC at specified IP address.

Viewing Live BIOS And Firmware Settings With `showlive`

The `showlive` command of cmsh displays the BIOS values currently used by the node. These are not necessarily the same as the currently configured values, because the currently configured values may not yet have been pushed to the node. The `showlive` command is available from within the dellsettings submode, which is under the node or category object. If live settings have not yet been fetched, then the operation can be forced with the `-f` option.

Example

```
[bright82->device[node001]]% showlive
[bright82->device[node001]]% showlive -f

Parameter Value
---------------- -------------------------
Readonly no
Revision
applyImmediately no
biosBootOrder NIC.Embedded.1-1-1, NIC.Mezzanine.1A-1,...
collaborativeCpuPerfCtrl no

© Bright Computing, Inc.
15.4 Frequently Asked Questions

1. Where are the log files for the Apply and Firmware update operations?

All Apply actions are logged in /var/log/cm-dell-manage.log on the head node, and all Firmware update actions are logged to /var/log/cm-dell-manage.log on the node itself.

2. Why does the Bright View display of the BIOS setting display nothing?

This is likely because the required information is not fetched using racadm when the node booted for the first time. To force this operation at the time of request, the showlive -f command from cmsh should be used (page 621). It is generally a good idea to run that command if unsure about the status of current and planned BIOS settings for the next boot.

Example

3. Why do the Bright View Dell settings for a node show no configured BIOS or NIC device settings?

This is because there are no Dell settings configured for a node. Settings can be initialized using cmsh with the importlive command. If there is no live information for the node yet, then the command importlive -f can be used:

Example
4. Why do the `showlive -f` or `importlive -f` commands fail, saying that the BMC user privileges are insufficient in the reports in `/var/log/cm-dell-manage.log`?

This is because the default BMC user `bright` with user id 4 does not have sufficient privileges to perform this operation. The default BMC user, user ID, and password that the CMDaemon uses for performing this operation, should be changed to a BMC user that has iDRAC management privileges. This can be done from the Settings window of the node or category in Bright View:

Devices→Nodes→Edit→Settings→BMC settings

Grouping→Node Categories→Settings→BMC settings
or from each node or base partition submode in cmsh.
This appendix contains lists of system configuration files that are managed by CMDaemon, and system configuration files that are managed by node-installer. These are files created or modified on the head nodes (section A.1), in the software images (section A.2), and on the regular nodes (section A.3). These files should not be confused with configuration files that are merely installed (section A.4).

Section 2.6.5 describes how system configuration files on all nodes are written out using the Cluster Management Daemon (CMDaemon). CMDaemon is introduced in section 2.6.5 and its configuration directives are listed in Appendix C.

All of these configuration files may be listed as Frozen Files in the CMDaemon configuration file to prevent them from being modified further by CMDaemon. The files can be frozen for the head node by setting the directive at /cm/local/apps/cmd/etc/cmd.conf. They can also be frozen on the regular nodes by setting the directive in the software image, by default at /cm/images/default-image/cm/local/apps/cmd/etc/cmd.conf.

A.1 System Configuration Files Created Or Modified By CMDaemon On Head Nodes

A list of CMDaemon-managed files on the head node can be seen by running the script /cm/local/apps/cmd/scripts/cm-list-managed-files on the head node. The head node files that are managed by CMDaemon are listed here for a plain installation on RHEL7 and derivatives. The list excludes the files managed by CMDaemon in the software images used for regular nodes (section A.2), which are by default located on the head node under /cm/images/:

Files generated or modified automatically on the head node by CMDaemon:
<table>
<thead>
<tr>
<th>File</th>
<th>Part</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>/etc/aliases</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/etc/dhclient.conf</td>
<td>Entire file</td>
<td>Red Hat only</td>
</tr>
<tr>
<td>/etc/dhcpd.conf</td>
<td>Entire file</td>
<td></td>
</tr>
<tr>
<td>/etc/dhcpd.internalnet.conf</td>
<td>Entire file</td>
<td>For internal networks other than internalnet, corresponding files are generated if node booting (table 3.2.1) is enabled</td>
</tr>
<tr>
<td>/etc/exports</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/etc/fstab</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/etc/genders</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/etc/hosts</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/etc/named.conf</td>
<td>Entire file</td>
<td>For zone additions use /etc/named.conf.include¹. For options additions, use /etc/named.conf.global.options.include</td>
</tr>
</tbody>
</table>
...continued

<table>
<thead>
<tr>
<th>File</th>
<th>Part</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>/etc/ntp.conf</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/etc/ntp/step-tickers</td>
<td>Section</td>
<td>Red Hat only</td>
</tr>
<tr>
<td>/etc/postfix/canonical</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/etc/postfix/generic</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/etc/postfix/main.cf</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/etc/resolv.conf</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/etc/shorewall/interfaces</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/etc/shorewall/masq</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/etc/shorewall/netmap</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/etc/shorewall/policy</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/etc/shorewall/zones</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/etc/shorewall6/interfaces</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/etc/shorewall6/netmap</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/etc/shorewall6/policy</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/etc/shorewall6/zones</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/etc/sysconfig/bmccfg</td>
<td>Entire file</td>
<td>BMC configuration</td>
</tr>
<tr>
<td>/etc/sysconfig/clock</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/etc/sysconfig/dhcpd</td>
<td>Entire file</td>
<td></td>
</tr>
<tr>
<td>/etc/sysconfig/network-scripts/ifcfg-*</td>
<td>Section</td>
<td>Red Hat only</td>
</tr>
<tr>
<td>/etc/sysconfig/network/routes</td>
<td>Section</td>
<td>SUSE only</td>
</tr>
<tr>
<td>/etc/sysconfig/network/ifcfg-*</td>
<td>Section</td>
<td>SUSE only</td>
</tr>
<tr>
<td>/etc/sysconfig/network/dhcp</td>
<td>Section</td>
<td>SUSE only</td>
</tr>
<tr>
<td>/tftpboot/mtu.conf</td>
<td>Entire file</td>
<td>Bright Computing configuration</td>
</tr>
<tr>
<td>/tftpboot/pxelinux.cfg/category.default</td>
<td>Entire file</td>
<td>Bright Computing configuration</td>
</tr>
<tr>
<td>/var/named/*.zone¹</td>
<td>Entire file</td>
<td>Red Hat only. For custom additions use /var/named/*.zone.include</td>
</tr>
<tr>
<td>/etc/HOSTNAME</td>
<td>Entire file</td>
<td>SUSE only</td>
</tr>
<tr>
<td>/var/lib/named/*.zone¹</td>
<td>Entire file</td>
<td>SUSE only. For custom additions use /var/lib/named/*.zone.include</td>
</tr>
</tbody>
</table>

¹ User-added zone files ending in *.zone that are placed for a corresponding zone statement in the include file /etc/named.conf.include are wiped by CMDaemon activity. Another pattern, eg: *.myzone, must therefore be used instead

A.2 System Configuration Files Created Or Modified By CMDaemon In Software Images:

The list of CMDaemon-managed system configuration files in the software images, that is, under /cm/images/<image>, can be seen by running cm-list-managed-files and filtering the list appropriately. For example using | grep /cm/images.

For the default image, under /cm/images/default-image, the list is:

System configuration files created or modified by CMDaemon in software images

© Bright Computing, Inc.
### A.3 Files Created On Regular Nodes By The Node-Installer

The list of files on the regular node that are generated or modified by the node-installer can be viewed in the logs at `/var/log/modified-by-node-installer.log` on the node. For a default installation the result is shown in the following table:

<table>
<thead>
<tr>
<th>File</th>
<th>Part</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>/etc/aliases</code></td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td><code>/etc/hosts</code></td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td><code>/etc/init/serial.conf</code></td>
<td>Section</td>
<td>only in RHEL6.x, not in RHEL7.x</td>
</tr>
<tr>
<td><code>/etc/inittab</code></td>
<td>Section</td>
<td>SUSE only</td>
</tr>
<tr>
<td><code>/etc/mkinitrd_cm.conf</code></td>
<td>Section</td>
<td>Red Hat only</td>
</tr>
<tr>
<td><code>/etc/modprobe.d/bright-cmdaemon.conf</code></td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td><code>/etc/postfix/main.cf</code></td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td><code>/etc/securetty</code></td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td><code>/etc/sysconfig/clock</code></td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td><code>/etc/sysconfig/init</code></td>
<td>Section</td>
<td>only in RHEL6.x, not in RHEL7.x</td>
</tr>
<tr>
<td><code>/etc/sysconfig/network/dhcp</code></td>
<td>Section</td>
<td>SUSE only</td>
</tr>
</tbody>
</table>
A.4 Files Not Generated, But Installed.

This appendix (Appendix A) is mainly about generated configuration files. This section (A.4) of the appendix discusses a class of files that is not generated, but may still be confused with generated files. The discussion in this section clarifies the issue, and explains how to check if non-generated installed files differ from the standard distribution installation.

A design goal of Bright Cluster Manager is that of minimal interference. That is, to stay out of the way of the distributions that it works with as much as is reasonable. Still, there are inevitably cluster manager configuration files that are not generated, but installed from a cluster manager package. A cluster manager configuration file of this kind overwrites the distribution configuration file with its own special settings to get the cluster running, and the file is then not maintained by the node-installer or CMDaemon. Such files are therefore not listed on any of the tables in this chapter.

Sometimes the cluster file version may differ unexpectedly from the distribution version. To look into this, the following steps may be followed:

Is the configuration file a Bright Cluster Manager version or a distribution version? A convenient way to check if a particular file is a cluster file version is to grep for it in the packages list for the cluster packages. For example, for nsswitch.conf:

<table>
<thead>
<tr>
<th>File</th>
<th>Part</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>/etc/sysconfig/network</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/etc/sysconfig/network/dhcp</td>
<td>Section</td>
<td>SUSE only</td>
</tr>
<tr>
<td>/etc/sysconfig/network/ifcfg-*</td>
<td>Entire file</td>
<td>SUSE only, not ifcfg-lo</td>
</tr>
<tr>
<td>/etc/sysconfig/network-scripts/ifcfg-*</td>
<td>Entire file</td>
<td>Red Hat only, not ifcfg-lo</td>
</tr>
<tr>
<td>/etc/resolv.conf</td>
<td>Entire file</td>
<td></td>
</tr>
<tr>
<td>/etc/hosts</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/etc/postfix/main.cf</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/etc/ntp.conf</td>
<td>Entire file</td>
<td></td>
</tr>
<tr>
<td>/etc/ntp/step-tickers</td>
<td>Entire file</td>
<td>Red Hat only</td>
</tr>
<tr>
<td>/etc/hostname</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/cm/local/apps/cmd/etc/cert.pem</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/cm/local/apps/cmd/etc/cert.key</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/cm/local/apps/openldap/etc/certs/ldap.pem</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/cm/local/apps/openldap/etc/certs/ldap.key</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/var/spool/cmd/disks.xml</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/var/log/node-installer</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/var/log/modified-by-node-installer.log</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>/etc/init/serial.conf</td>
<td>Section</td>
<td>Red Hat only, not in RHEL7.x</td>
</tr>
<tr>
<td>/etc/inittab</td>
<td>Section</td>
<td>SUSE only</td>
</tr>
<tr>
<td>/etc/mkinitrd_cm.conf</td>
<td>Section</td>
<td>Red Hat only</td>
</tr>
</tbody>
</table>
The inner `repoquery` displays a list of all the packages. By grepping for the cluster manager version string, for example `_cm8.2` for Bright Cluster Manager 8.2, the list of cluster manager packages is found. The outer `repoquery` displays the list of files within each package in the list of cluster manager packages. By grepping for `nsswitch.conf$, any file paths ending in `nsswitch.conf` in the cluster manager packages are displayed. The output is:

```
/cm/conf/etc/nsswitch.conf
/cm/conf/etc/nsswitch.conf
/cm/node-installer/etc/nsswitch.conf
```

Files under `/cm/conf` are placed by Bright Cluster Manager packages when updating the head node. From there they are copied over during the post-install section of the RPM to where the distribution version configuration files are located by the cluster manager, but only during the initial installation of the cluster. The distribution version file is overwritten in this way to prevent RPM dependency conflicts of the Bright Cluster Manager version with the distribution version. The configuration files are not copied over from `/cm/conf` during subsequent reboots after the initial installation. The `cm/conf` files are however updated when the Bright Cluster Manager packages are updated. During such a Bright Cluster Manager update, a notification is displayed that new configuration files are available.

Inverting the cluster manager version string match displays the files not provided by Bright Cluster Manager. These are normally the files provided by the distribution:

```
[root@bright82 ~]# repoquery -l $(repoquery -a | grep -F -v _cm8.2) | grep nsswitch.conf$

/etc/nsswitch.conf
/etc/nsswitch.conf
/usr/share/doc/yp-tools-2.9/nsswitch.conf
```

Which package provides the file in Bright Cluster Manager and in the distribution? The packages that provide these files can be found by running the "`yum whatprovides *`" command on the paths given by the preceding output, for example:

```
"#/yum whatprovides */cm/conf/etc/nsswitch.conf
```

This reveals that some Bright Cluster Manager LDAP packages can provide an `nsswitch.conf` file. The file is a plain file provided by the unpacking and placement that takes place when the package is installed. The file is not generated or maintained periodically after placement, which is the reason why this file is not seen in the tables of sections A.1, A.2, and A.3 of this appendix.

Similarly, looking through the output for

```
"#/yum whatprovides */etc/nsswitch.conf
```

shows that `glibc` provides the distribution version of the `nsswitch.conf` file, and that there is also a node-installer version of this file available from the Bright Cluster Manager packages.

What are the differences between the Bright Cluster Manager version and the distribution versions of the file? Sometimes it is helpful to compare a distribution version and cluster version of `nsswitch.conf` to show the differences in configuration. The versions of the RPM packages containing the `nsswitch.conf` can be downloaded, their contents extracted, and their differences compared as follows:

```
"# mkdir yumextracted ; cd yumextracted
"# yumdownloader glibc-2.12-1.107.el6.x86_64.rpm
"# rpm2cpio glibc-2.12-1.107.el6.x86_64.rpm | cpio -idmv
"# yumdownloader cm-config-ldap-client-6.0-45_cm8.2.noarch.rpm
"# rpm2cpio cm-config-ldap-client-6.0-45_cm8.2.noarch.rpm | cpio -idmv
"# diff etc/nsswitch.conf cm/conf/etc/nsswitch.conf
```

© Bright Computing, Inc.
**What are the configuration files in an RPM package?**  An RPM package allows files within it to be marked as configuration files. Files marked as configuration files can be listed with `rpm -qc <package>`. Optionally, piping the list through “sort -u” filters out duplicates.

**Example**

```bash
  "# rpm -qc glibc | sort -u
  /etc/ld.so.cache
  /etc/ld.so.conf
  /etc/localtime
  /etc/nsswitch.conf
  /usr/lib64/gconv/gconv-modules
  /var/cache/ldconfig/aux-cache
```

**How does an RPM installation deal with local configuration changes? Are there configuration files or critical files that Bright Cluster Manager misses?**  Whenever an RPM installation detects a file with local changes, it can treat the local system file as if:

1. the local system file is frozen\(^1\). The installation does not interfere with the local file, but places the updated file as an .rpmnew file in the same directory.
2. the local system file is not frozen. The installation changes the local file. It copies the local file to an .rpmsave file in the same directory, and installs a new file from the RPM package.

When building the Bright Cluster Manager packages, the package builders can specify which of these two methods apply. When dealing with the built package, the system administrator can use an `rpm` query method to determine which of the two methods applies for a particular file in the package. For example, for glibc, the following query can be used and grepped:

```bash
  rpm -q --queryformat '\[%{FILENAMES}\t%{FILEFLAGS:fflags}\n\]' glibc | egrep '\[[:space:]]\.(c|n).*$' | sort -u
  /etc/ld.so.cache cmng
  /etc/ld.so.conf cn
  /etc/localtime cn
  /etc/nsswitch.conf cn
  /usr/lib64/gconv/gconv-modules cn
  /usr/lib/gconv/gconv-modules cn
  /var/cache/ldconfig/aux-cache cmng
```

Here, the second column of the output displayed shows which of the files in the package have a configuration (c) flag or a noreplace (n) flag. The c flag without the n flag indicates that an .rpmsave file will be created, while a c flag together with an n flag indicates that an .rpmnew file will be created.

In any case, files that are not marked as configuration files are overwritten during installation. So:

- If a file is not marked as a configuration file, and it has been customized by the system administrator, and this file is provided by an RPM package, and the RPM package is updated on the system, then the file is overwritten silently.
- If a file is marked as a configuration file, and it has been customized by the system administrator, and this file is provided by an RPM package, and the RPM package is updated on the system, then it is good practice to look for .rpmsave and .rpmnew versions of that file, and run a comparison on detection.

---

\(^1\)This freezing should not be confused with the `FrozenFile` directive (Appendix C), where the file or section of a file is being maintained by CMDaemon, and where freezing the file prevents CMDaemon from maintaining it.
Bright Cluster Manager should however mark all critical files as configuration files in the Bright Cluster Manager packages.

Sometimes, RPM updates can overwrite a particular file that the administrator has changed locally and then would like to keep frozen.

To confirm that this is the problem, the following should be checked:

- The `--queryformat` option should be used to check that file can indeed be overwritten by updates. If the file has an `n` flag (regardless of whether it is a configuration file or not) then overwriting due to RPM updates does not happen, and the local file remains frozen. If the file has no `n` flag, then replacement occurs during RPM updates.

For files with no `n` flag, but where the administrator would still like to freeze the file during updates, the following can be considered:

- The file text content should be checked to see if it is a CMDaemon-maintained file (section 2.6.5), or checked against the list of generated files (Appendix A). This is just to make sure to avoid confusion about how changes are occurring in such a file.
  
  - If it is a CMDaemon-maintained file, then configuration changes put in by the administrator will also not persist in the maintained section of the file unless the `FrozenFile` directive (section C) is used to freeze the change.
  
  - If it is only a section that CMDaemon maintains, then configuration changes can be placed outside of the maintained section.

Wherever the changes are placed in such a file, these changes are in any case by default overwritten on RPM updates if the file has no `n` flag.

- Some regular node updates can effectively be maintained in a desired state with the help of a finalize script (Appendix E).

- Updates can be excluded from YUM/zypper (section 11.3.2), thereby avoiding the overwriting of that file by the excluded package.

A request to change the package build flag may be sent to Bright Computing if the preceding suggested options are unworkable.
CMDaemon Configuration File Directives

This appendix lists all configuration file directives that may be used in the cluster management daemon configuration file. If a change is needed, then the directives are normally changed on the head node, or on both head nodes in the high availability configuration, in:

/cm/local/apps/cmd/etc/cmd.conf

The directives can also be set in some cases for the regular nodes, via the software image in /cm/images/default-image/cm/local/apps/cmd/etc/cmd.conf on the head node. Changing the defaults already there is however not usually needed, and is not recommended.

Only one directive is valid per cmd.conf file.
To activate changes in a cmd.conf configuration file, the cmd service associated with it must be restarted.

- For the head node this is normally done with the command:

  service cmd restart

- For regular nodes, cmd running on the nodes is restarted. Often, the image should be updated before cmd is restarted. How to carry out these procedures for a directive is described with an example where the FrozenFile directive is activated on a regular node on page 645.

Master directive

**Syntax:** Master = hostname  
**Default:** Master = master

The cluster management daemon treats the host specified in the Master directive as the head node. A cluster management daemon running on a node specified as the head node starts in head mode. On a regular node, it starts in node mode.

Port directive

**Syntax:** Port = number  
**Default:** Port = 8080

The number used in the syntax above is a number between 0 and 65535. The default value is 8080.

The Port directive sets the value of the port of the cluster management daemon to listen for non-SSL HTTP calls. By default, this happens only during init. All other communication with the cluster
management daemon is carried out over the SSL port. Pre-init port adjustment can be carried out in the node-installer.conf configuration. Shorewall may need to be modified to allow traffic through for a changed port.

**SSLPort directive**

**Syntax:** SSLPort = number

**Default:** SSLPort = 8081

The number used in the syntax above is a number between 0 and 65535. The default value is 8081.

The SSLPort directive sets the value of the SSL port of the cluster management daemon to listen for SSL HTTP calls. By default, it is used for all communication of CMDaemon with Bright View and cmsh, except for when CMDaemon is started up from init.

**SSLPortOnly directive**

**Syntax:** SSLPortOnly = yes | no

**Default:** SSLPortOnly = no

The SSLPortOnly directive allows the non-SSL port to be disabled. During normal running, both SSL and non-SSL ports are listening, but only the SSL port is used. By default, the non-SSL port is only used during CMDaemon start up.

If PXEBootOverHTTP (page 656) is set to 1, then SSLPortOnly must be set to no.

**CertificateFile directive**

**Syntax:** CertificateFile = filename

**Default:** CertificateFile = "/cm/local/apps/cmd/etc/cert.pem"

The CertificateFile directive specifies the PEM-format certificate which is to be used for authentication purposes. On the head node, the certificate used also serves as a software license.

**PrivateKeyFile directive**

**Syntax:** PrivateKeyFile = filename

**Default:** PrivateKeyFile = "/cm/local/apps/cmd/etc/cert.key"

The PrivateKeyFile directive specifies the PEM-format private key which corresponds to the certificate that is being used.

**CACertificateFile directive**

**Syntax:** CACertificateFile = filename

**Default:** CACertificateFile = "/cm/local/apps/cmd/etc/cacert.pem"

The CACertificateFile directive specifies the path to the Bright Cluster Manager PEM-format root certificate. It is normally not necessary to change the root certificate.

**ClusterCertificateFile directive**

**Syntax:** ClusterCertificateFile = filename

**Default:** ClusterCertificateFile = "/cm/local/apps/cmd/etc/cluster.pem"
The `ClusterCertificateFile` directive specifies the path to the Bright Cluster Manager PEM-format cluster certificate file used as a software license, and to sign all client certificates.

**ClusterPrivateKeyFile directive**

*Syntax:* `ClusterPrivateKeyFile = filename`
*Default:* `ClusterPrivateKeyFile = "/cm/local/apps/cmd/etc/cluster.key"

The `ClusterPrivateKeyFile` directive specifies the path to the Bright Cluster Manager PEM-format private key which corresponds to the cluster certificate file.

**RandomSeedFile directive**

*Syntax:* `RandomSeedFile = filename`
*Default:* `RandomSeedFile = "/dev/urandom"

The `RandomSeedFile` directive specifies the path to a source of randomness.

**DHParamFile directive**

*Syntax:* `DHParamFile = filename`
*Default:* `DHParamFile = "/cm/local/apps/cmd/etc/dh1024.pem"

The `DHParamFile` directive specifies the path to the Diffie-Hellman parameters.

**SSLHandshakeTimeout directive**

*Syntax:* `SSLHandshakeTimeout = number`
*Default:* `SSLHandshakeTimeout = 10`

The `SSLHandshakeTimeout` directive controls the time-out period (in seconds) for SSL handshakes.

**SSLSessionCacheExpirationTime directive**

*Syntax:* `SSLSessionCacheExpirationTime = number`
*Default:* `SSLSessionCacheExpirationTime = 300`

The `SSLSessionCacheExpirationTime` directive controls the period (in seconds) for which SSL sessions are cached. Specifying the value 0 can be used to disable SSL session caching.

**DBHost directive**

*Syntax:* `DBHost = hostname`
*Default:* `DBHost = "localhost"

The `DBHost` directive specifies the hostname of the MySQL database server.

**DBPort directive**

*Syntax:* `DBPort = number`
*Default:* `DBPort = 3306`

The `DBPort` directive specifies the TCP port of the MySQL database server.
DBUser directive

Syntax: DBUser = username
Default: DBUser = cmdaemon

The DBUser directive specifies the username used to connect to the MySQL database server.

DBPass directive

Syntax: DBPass = password
Default: DBPass = "<random string set during installation>"

The DBPass directive specifies the password used to connect to the MySQL database server.

DBName directive

Syntax: DBName = database
Default: DBName = "cmdaemon"

The DBName directive specifies the database used on the MySQL database server to store CMDaemon related configuration and status information.

DBUnixSocket directive

Syntax: DBUnixSocket = filename
Default: DBUnixSocket = "/var/lib/mysql/mysql.sock"

The DBUnixSocket directive specifies the named pipe used to connect to the MySQL database server if it is running on the same machine.

DBUpdateFile directive

Syntax: DBUpdateFile = filename
Default: DBUpdateFile = "/cm/local/apps/cmd/etc/cmdaemon_upgrade.sql"

The DBUpdateFile directive specifies the path to the file that contains information on how to upgrade the database from one revision to another.

EventBucket directive

Syntax: EventBucket = filename
Default: EventBucket = "/var/spool/cmd/eventbucket"

The EventBucket directive (section 12.10.3) specifies the path to the named pipe that is created to listen for incoming events from a user.

EventBucketFilter directive

Syntax: EventBucketFilter = filename
Default: EventBucketFilter = "/cm/local/apps/cmd/etc/eventbucket.filter"

The EventBucketFilter directive (section 12.10.3) specifies the path to the file that contains regular expressions used to filter out incoming messages on the event-bucket.
LDAPHost directive
Syntax: LDAPHost = hostname
Default: LDAPHost = "localhost"

The LDAPHost directive specifies the hostname of the LDAP server to connect to for user management.

LDAPUser directive
Syntax: LDAPUser = username
Default: LDAPUser = "root"

The LDAPUser directive specifies the username used when connecting to the LDAP server.

LDAPPass directive
Syntax: LDAPPass = password
Default: LDAPPass = "<random string set during installation>"

The LDAPPass directive specifies the password used when connecting to the LDAP server. It can be changed following the procedure described in Appendix I.

LDAPReadOnlyUser directive
Syntax: LDAPReadOnlyUser = username
Default: LDAPReadOnlyUser = "readonlyroot"

The LDAPReadOnlyUser directive specifies the username that will be used when connecting to the LDAP server during LDAP replication. The user is a member of the "rogroup" group, whose members have a read-only access to the whole LDAP directory.

LDAPReadOnlyPass directive
Syntax: LDAPReadOnlyPass = password
Default: LDAPReadOnlyPass = "<random string set during installation>"

The LDAPReadOnlyPass directive specifies the password that will be used when connecting to the LDAP server during LDAP replication.

LDAPSearchDN directive
Syntax: LDAPSearchDN = dn
Default: LDAPSearchDN = "dc=cm,dc=cluster"

The LDAPSearchDN directive specifies the Distinguished Name (DN) used when querying the LDAP server.

HomeRoot directive
Syntax: HomeRoot = path
Default: HomeRoot = "/home"

The HomeRoot directive specifies the default user home directory used by CMDaemon. It is used for automatic mounts, exports, and when creating new users.
DocumentRoot directive
Syntax: DocumentRoot = path
Default: DocumentRoot = "/cm/local/apps/cmd/etc/htdocs"

The DocumentRoot directive specifies the directory mapped to the web-root of the CMDaemon. The CMDaemon acts as a HTTP-server, and can therefore in principle also be accessed by web-browsers.

SpoolDir directive
Syntax: SpoolDir = path
Default: SpoolDir = "/var/spool/cmd"

The SpoolDir directive specifies the directory which is used by the CMDaemon to store temporary and semi-temporary files.

EnableJSON directive
Syntax: EnableJSON = true | false
Default: EnableJSON = true

The EnableJSON directive allows Bright View. If the administrator or other user create other JSON-dependent utilities, these require the directive to be true too.

EnableWebSocketService directive
Syntax: EnableWebSocketService = true | false
Default: EnableWebSocketService = true

The EnableWebSocketService directive allows the use of CMDaemon Lite (section 2.6.7).

EnablePrometheusMetricService directive
Syntax: EnablePrometheusMetricService = true | false
Default: EnablePrometheusMetricService = true

If true, the EnablePrometheusMetricService directive enables monitoring plots in Bright View and the User Portal.

PrometheusMetricServicePath directive
Syntax: PrometheusMetricServicePath = path
Default: PrometheusMetricServicePath = SCRIPTS_DIR"metrics/prometheus"

The PrometheusMetricServicePath directive is the path from which CMDaemon can serve Prometheus metrics. SCRIPTS_DIR is the stem path /cm/local/apps/cmd/scripts by default.

CMDaemonAudit directive
Syntax: CMDaemonAudit = yes | no
Default: CMDaemonAudit = no

When the CMDaemonAudit directive is set to yes, and a value is set for the CMDaemon auditor file with
the CMDaemonAuditorFile directive, then CMDaemon actions are time-stamped and logged in the CMDaemon auditor file.

**CMDaemonAuditorFile directive**

**Syntax:** CMDaemonAuditorFile = *filename*

**Default:** CMDaemonAuditorFile = "/var/spool/cmd/audit.log"

The CMDaemonAuditorFile directive sets where the audit logs for CMDaemon actions are logged. The log format is:

(time stamp) profile [IP-address] action (unique key)

**Example**

(Mon Jan 31 12:41:37 2011) Administrator [127.0.0.1] added Profile: arbitprof(4294967301)

**DisableAuditorForProfiles directive**

**Syntax:** DisableAuditorForProfiles = {profile}...

**Default:** DisableAuditorForProfiles = {node}

The DisableAuditorForProfiles directive sets the profile for which an audit log for CMDaemon actions is disabled. A profile (section 2.3.4) defines the services that CMDaemon provides for that profile user. More than one profile can be set as a comma-separated list. Out of the profiles that are available on a newly-installed system: node, admin, cmhealth, and readonly; only the profile node is enabled by default. New profiles can also be created via the profile mode of cmsh or via the clickpath Identity Management—Profiles— of Bright View, thus making it possible to disable auditing for arbitrary groups of CMDaemon services.

**EventLogger directive**

**Syntax:** EventLogger = true|false

**Default:** EventLogger = false

The EventLogger directive sets whether to log events. If active, then by default it logs events to /var/spool/cmd/events.log on the active head. If a failover takes place, then the event logs on both heads should be checked and merged for a complete list of events.

The location of the event log on the filesystem can be changed using the EventLoggerFile directive (page 641).

Whether events are logged in files or not, events are cached and accessible using cmsh. The number of events cached by CMDaemon is determined by the parameter MaxEventHistory (page 641).

**EventLoggerFile directive**

**Syntax:** EventLoggerFile = *filename*

**Default:** EventLogger = "/var/spool/cmd/events.log"

The EventLogger directive sets where the events seen in the event viewer (section 12.10.1) are logged.

**MaxEventHistory directive**

**Syntax:** AdvancedConfig = {"MaxEventHistory=number", ...}

**Default:** MaxEventHistory=8192

MaxEventHistory is a parameter of the AdvancedConfig (page 649) directive.

© Bright Computing, Inc.
By default, when not explicitly set, the maximum number of events that is retained by CMDaemon is 8192. Older events are discarded.

The parameter can take a value from 0 to 1000000. However, CMDaemon is less responsive with larger values, so in that case, setting the EventLogger directive (page 641) to true, to activate logging to a file, is advised instead.

**PublicDNS directive**

**Syntax:** PublicDNS = true|false  
**Default:** PublicDNS = false

By default, internal hosts are resolved only if requests are from the internal network. Setting PublicDNS to true allows the head node name server to resolve internal network hosts for any network, including networks that are on other interfaces on the head node. Separate from this directive, port 53/UDP must also be opened up in Shorewall (section 7.2 of the Installation Manual) if DNS is to be provided for queries from an external network.

**LockDownDhcpd directive**

**Syntax:** LockDownDhcpd = true|false  
**Default:** LockDownDhcpd = false

LockDownDhcpd is a deprecated legacy directive. If set to true, a global DHCP “deny unknown-clients” option is set. This means no new DHCP leases are granted to unknown clients for all networks. Unknown clients are nodes for which Bright Cluster Manager has no MAC addresses associated with the node. The directive LockDownDhcpd is deprecated because its globality affects clients on all networks managed by Bright Cluster Manager, which is contrary to the general principle of segregating the network activity of networks.

The recommended way now to deny letting new nodes boot up is to set the option for specific networks by using cmsh or Bright View (section 3.2.1, figure 3.5, table 3.2.1). Setting the cmd.conf LockDownDhcpd directive overrides lockdowndhcpd values set by cmsh or Bright View.

**MaxNumberOfProvisioningThreads directive**

**Syntax:** MaxNumberOfProvisioningThreads = number  
**Default:** MaxNumberOfProvisioningThreads = 10000

The MaxNumberOfProvisioningThreads directive specifies the cluster-wide total number of nodes that can be provisioned simultaneously. Individual provisioning servers typically define a much lower bound on the number of nodes that may be provisioned simultaneously.

**SetupBMC directive**

**Syntax:** SetupBMC = true|false  
**Default:** SetupBMC = true

Configure the username and password for the BMC interface of the head node and regular nodes automatically. (This should not be confused with the setupBmc field of the node-installer configuration file, described in section 5.8.7.)

**BMCSessionTimeout directive**

**Syntax:** BMCSessionTimeout = number
Default: BMCSessionTimeout = 2000

The BMCSessionTimeout specifies the time-out for BMC calls in milliseconds.

**BMCIdentifyScript directive**

Syntax: AdvancedConfig = {"BMCIdentify=filename", ...}

Default: unset

BMCIdentifyScript is a parameter of the AdvancedConfig (page 649) directive. The parameter takes a full file path to a script that can be used for identification with a BMC (section 3.7.4).

**BMCIdentifyScriptTimeout directive**

Syntax: AdvancedConfig = {"BMCIdentifyScriptTimeout=number from 1 to 360", ...}

Default: 60

BMCIdentifyScriptTimeout is a parameter of the AdvancedConfig (page 649) directive. CMDaemon waits at the most BMCIdentifyScriptTimeout seconds for the script used by the BMCIdentify directive to complete.

**BMCIdentifyCache directive**

Syntax: AdvancedConfig = {"BMCIdentifyCache=0|1", ...}

Default: 1

BMCIdentifyCache is a parameter of the AdvancedConfig (page 649) directive. If set to 1, then CMDaemon remembers the last value of the output of the script used by the BMCIdentify directive.

**SnmpSessionTimeout directive**

Syntax: SnmpSessionTimeout = number

Default: SnmpSessionTimeout = 500000

The SnmpSessionTimeout specifies the time-out for SNMP calls in microseconds.

**PowerOffPDUOutlet directive**

Syntax: PowerOffPDUOutlet = true|false

Default: PowerOffPDUOutlet = false

Enabling the PowerOffPDUOutlet directive allows PDU ports to be powered off for clusters that have both PDU and IPMI power control. Section 4.1.3 has more on this.

**PowerThreadPoolSize directive**

Syntax: AdvancedConfig = {"PowerThreadPoolSize=<integer>", ...}

Default: PowerThreadPoolSize=32

PowerThreadPoolSize is a parameter of the AdvancedConfig (page 649) directive. The parameter can take positive integer values. Increasing its value increases the number of threads that are used to power up the nodes in a cluster (section 4.2.3), so that the cluster is fully operational quicker. The administrator should however take into account the power surge due to increasing the number of threads (number of subprocesses) before increasing the value beyond its default.

© Bright Computing, Inc.
**DisableBootLogo directive**

Syntax: DisableBootLogo = true | false  
Default: DisableBootLogo = false

When DisableBootLogo is set to true, the Bright Cluster Manager logo is not displayed on the first boot menu when nodes PXE boot.

**StoreBIOSTimeInUTC directive**

Syntax: StoreBIOSTimeInUTC = true | false  
Default: StoreBIOSTimeInUTC = false

When StoreBIOSTimeInUTC is set to true, the system relies on the time being stored in BIOS as being UTC rather than local time.

**FreezeChangesTo<wlm>Config directives:**

* FreezeChangesToCondorConfig directive
* FreezeChangesToPBSPro directive
* FreezeChangesToSGEConfig directive
* FreezeChangesToUGEConfig directive
* FreezeChangesToSlurmConfig directive
* FreezeChangesToLSFConfig directive

Syntax: FreezeChangesTo<wlm>Config = true | false  
Default: FreezeChangesTo<wlm>Config = false

When FreezeChangesTo<wlm>Config is set to true, the CMDaemon running on that node does not make any modifications to the workload manager configuration for that node. Workload managers for which this value can be set are:

- Condor
- PBSPro
- SGE
- UGE
- Slurm
- LSF

Monitoring of jobs, and workload accounting and reporting continues for frozen workload managers.

Upgrades to newer workload manager versions may still require some manual adjustments of the configuration file, typically if a newer version of the workload manager configuration changes the syntax of one of the options in the file.

**FrozenFile directive**

Syntax: FrozenFile = { filename[, filename]... }  
Example: FrozenFile = { "/etc/dhcpd.conf", "/etc/postfix/main.cf" }

The FrozenFile directive is used to prevent the CMDaemon-maintained sections of configuration files
from being automatically generated. This is useful when site-specific modifications to configuration files have to be made.

To avoid problems, the file that is frozen should not be a symlink, but should be the ultimate destination file. The `readlink -f <symlinkname>` command returns the ultimate destination file of a symlink called `<symlinkname>`. This is also the case for an ultimate destination file that is reached via several chained symlinks.

**FrozenFile directive for regular nodes**

**FrozenFile directive for regular nodes for CMDaemon**
The `FrozenFile` directive can be used within the `cmd.conf` file of the regular node.

**Example**

To freeze the file `/etc/named.conf` on the regular nodes running with the image `default-image`, the file:

```
/cm/images/default-image/cm/local/apps/cmd/etc/cmd.conf
```
can have the following directive set in it:

```
FrozenFile = { "/etc/named.conf" }
```

The path of the file that is to be frozen on the regular node must be specified relative to the root of the regular node.

The running node should then have its image updated. This can be done with the `imageupdate` command in `cmsh` (section 5.6.2), or the `Update node` button in Bright View (section 5.6.3). After the update, CMDaemon should be restarted within that category of nodes:

**Example**

```
[root@bright82 ~]# pdsh -v -g category=default service cmd restart
node002: Waiting for CMDaemon (25129) to terminate...
node001: Waiting for CMDaemon (19645) to terminate...
node002: [ OK ]
node001: [ OK ]
node002: Waiting for CMDaemon to start...[ OK ]
node001: Waiting for CMDaemon to start...[ OK ]
```

**FrozenFile directive for regular nodes for the node-installer**

CMDaemon directives only affect files on a regular node after CMDaemon starts up on the node during the init stage. So files frozen by the CMDaemon directive stay unchanged by CMDaemon after this stage, but they may still be changed before this stage.

Freezing files so that they also stay unchanged during the pre-init stage—that is during the node-installer stage—is possible with node-installer directives.

Node-installer freezing is independent of CMDaemon freezing, which means that if a file freeze is needed for the entire startup process as well as beyond, then both a node-installer as well as a CMDaemon freeze are sometimes needed.

Node-installer freezes can be done with the node-installer directives in `/cm/node-installer/scripts/node-installer.conf`, introduced in section 5.4:

- `frozenFilesPerNode`
- `frozenFilesPerCategory`

**Example**

© Bright Computing, Inc.
Per node:

\[
\text{frozenFilesPerNode} = "*:\text{/localdisk/etc/ntp.conf}"; "\text{node003:\text{/localdisk/etc/hosts}}"
\]

Here, the * wildcard means that no restriction is set. Setting node003 means that only node003 is frozen.

**Example**

Per category:

\[
\text{frozenFilesPerCategory} = "\text{mycategory:\text{/localdisk/etc/sysconfig/network-scripts/ifcfg-eth1}}"
\]

Here, the nodes in the category mycategory are prevented from being changed by the node-installer.

**The Necessity Of A FrozenFile Directive**

In a configuration file after a node is fully up, the effect of a statement earlier on can often be overridden by a statement later in the file. So, the following useful behavior is independent of whether FrozenFile is being used for a configuration file or not: A configuration file, for example /etc/postfix/main.cf, with a configuration statement in an earlier CMDaemon-maintained part of the file, for example:

\[
\text{mydomain} = \text{eth.cluster}
\]

can often be overridden by a statement later on outside the CMDaemon-maintained part of the file:

\[
\text{mydomain} = \text{eth.gig.cluster}
\]

Using FrozenFile in CMDaemon or the node-installer can thus sometimes be avoided by the use of such overriding statements later on.

Whether overriding later on is possible depends on the software being configured. It is true for Postfix configuration files, for example, but it may not be so for the configuration files of other applications.

**EaseNetworkValidation directive**

**Syntax:** EaseNetworkValidation = 0 | 1 | 2

**Default:** EaseNetworkValidation = 0

CMDaemon enforces certain requirements on network interfaces and management/node-booting networks by default. In heavily customized setups, such as is common in Type 3 networks (section 3.3.7 of the Installation Manual), the user may wish to disable these requirements.

- 0 enforces all requirements.
- 1 allows violation of the requirements, with validation warnings. This value should never be set except under instructions from Bright support.
- 2 allows violation of the requirements, without validation warnings. This value should never be set except under instructions from Bright support.

**CustomUpdateConfigFileScript directive**

**Syntax:** CustomUpdateConfigFileScript = filename

**Default:** commented out in the default cmd.conf file

Whenever one or more entities have changed, the custom script at filename, specified by a full path, is called 30s later. Python bindings can be used to get information on the current setup.
**ConfigDumpPath directive**

*Syntax:* ConfigDumpPath = *filename*

*Default:* ConfigDumpPath = /var/spool/cmd/cmdaemon.config.dump

The ConfigDumpPath directive sets a dump file for dumping the configuration used by the power control script /cm/local/apps/cmd/scripts/pctl/pctl. The pctl script is a fallback script to allow power operations if CMDaemon is not running.

- If no directive is set (ConfigDumpPath = ""), then no dump is done.
- If a directive is set, then the administrator must match the variable cmdconfigfile in the powercontrol configuration file /cm/local/apps/cmd/scripts/pctl/config.py to the value of ConfigDumpPath. By default, the value of cmdconfigfile is set to /var/spool/cmd/cmdaemon.config.dump.

**SyslogHost directive**

*Syntax:* SyslogHost = *hostname*

*Default:* SyslogHost = "localhost"

The SyslogHost directive specifies the hostname of the syslog host.

**SyslogFacility directive**

*Syntax:* SyslogFacility = *facility*

*Default:* SyslogFacility = "LOG_LOCAL6"

The default value of LOG_LOCAL6 is set in:

- /etc/syslog.conf in Red Hat 5 and variants
- /etc/rsyslog.conf in Red Hat 6 and variants
- /etc/syslog-ng/syslog-ng.conf in SLES versions

These are the configuration files for the default syslog daemons syslog, rsyslog, and syslog-ng, respectively, that come with the distribution. Bright Cluster Manager redirects messages from CMDaemon to /var/log/cmdaemon only for the default syslog daemon that the distribution provides. So, if another syslog daemon other than the default is used, then the administrator has to configure the non-default syslog daemon facilities manually.

The value of facility must be one of: LOG_KERN, LOG_USER, LOG_MAIL, LOG_DAEMON, LOG_AUTH, LOG_SYSLOG or LOG_LOCAL0..7

**NameServerLocalhostLocation directive**

*Syntax:* AdvancedConfig = {"NameServerLocalhostLocation=0|1", ...}

*Default:* NameServerLocalhostLocation=0

NameServerLocalhostLocation is a parameter of the AdvancedConfig (page 649) directive.

When set to 1, the location of the localhost as specified by the nameserver directive in /etc/resolv.conf is moved to the bottom of the list of nameserver entries. The default value of 0 places it at the top of those entries.
ResolveToExternalName directive

Syntax: ResolveToExternalName = true | false
Default: ResolveToExternalName = false

The value of the ResolveToExternalName directive determines under which domain name the primary and secondary head node hostnames are visible from within the head nodes, and to which IP addresses their hostnames are resolved. Enabling this directive resolves the head nodes’ hostnames to the IP addresses of their external interfaces.

Thus, on head nodes and regular nodes in both single-head and failover clusters

- with ResolveToExternalName disabled, the master hostname and the actual hostname of the head node (e.g. head1, head2) by default always resolve to the internal IP address of the head node.
- with ResolveToExternalName enabled, the master hostname and the actual hostname of the head node (e.g. head1, head2) by default always resolve to the external IP address of the head node.

The resolution behavior can be summarized by the following table:

<table>
<thead>
<tr>
<th>ResolveToExternalName Directive Effects</th>
</tr>
</thead>
<tbody>
<tr>
<td>on simple head, on failover head, on regular node, Using the DNS?</td>
</tr>
<tr>
<td>master, resolving:</td>
</tr>
<tr>
<td>head</td>
</tr>
<tr>
<td>---------</td>
</tr>
<tr>
<td>ResolveToExternalName = False</td>
</tr>
<tr>
<td>I</td>
</tr>
<tr>
<td>ResolveToExternalName = True</td>
</tr>
<tr>
<td>E</td>
</tr>
<tr>
<td>E</td>
</tr>
</tbody>
</table>

Key: I: resolves to internal IP address of head
E: resolves to external IP address of head

The system configuration files on the head nodes that get affected by this directive include /etc/hosts and, on SLES systems, also the /etc/HOSTNAME. Also, the DNS zone configuration files get affected.

Additionally, in both the single-head and failover clusters, using the "hostname -f" command on a head node while ResolveToExternalName is enabled results in the host's Fully Qualified Domain Name (FQDN) being returned with the host's external domain name. That is, the domain name of the network that is specified as the "External network" in the base partition in cmsh (the output of "cmsh -c "partition use base; get externalnetwork").

Modifying the value of the ResolveToExternalName directive and restarting the CMDaemon while important system services (e.g. Slurm) are running should not be done. Doing so is likely to cause problems with accessing such services due to them then running with a different domain name than the one with which they originally started.

On a tangential note that is closely, but not directly related to the ResolveToExternalName directive: the cluster can be configured so that the "hostname -f" command executed on a regular node returns the FQDN of that node, and so that the FQDN in turn resolves to an external IP for that regular node. The details on how to do this are in the Bright Cluster Manager Knowledge Base at http://kb.brightcomputing.com/. A search query for FQDN leads to the relevant entry.
ResolveMasterToExternalName directive

Syntax: AdvancedConfig = \{ "ResolveMasterToExternalName=0|1", ... \}
Default: ResolveMasterToExternalName = 1

ResolveMasterToExternalName is a parameter of the AdvancedConfig (page 649) directive.
ResolveMasterToExternalName can only be used if ResolveToExternalName is active.
If set to 1 (the default), then the master head node, as specified by the name master, resolves to the IP address as set by ResolveToExternalName.
If set to 0 then the master head node resolves to the internal shared IP address.

ResolveMasterToExternalDomainName directive

Syntax: AdvancedConfig = \{ "ResolveMasterToExternalDomainName=0|1", ... \}
Default: ResolveMasterToExternalDomainName = 1

ResolveMasterToExternalDomainName is a parameter of the AdvancedConfig (page 649) directive.
ResolveMasterToExternalDomainName can only be used if ResolveToExternalName is active.
The external domain name as defined for the external network, in the form master.<external domain>, can be used in /etc/hosts during name resolution.
If set to 1 (the default), then the external domain name is used in /etc/hosts.
If set to 0, then the external domain name is not used in /etc/hosts.
Resolution occurs as shown in the following table:

<table>
<thead>
<tr>
<th>on simple head, resolving:</th>
<th>on failover head, resolving:</th>
<th>on regular node, resolving:</th>
<th>Using the DNS?</th>
</tr>
</thead>
<tbody>
<tr>
<td>master head</td>
<td>master head1 head2</td>
<td>master head(s)</td>
<td></td>
</tr>
<tr>
<td>ResolveToExternalName = False</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>I</td>
<td>I</td>
<td>I</td>
<td>I</td>
</tr>
<tr>
<td>I</td>
<td>I</td>
<td>I</td>
<td>I</td>
</tr>
<tr>
<td>ResolveToExternalName = True</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>E</td>
<td>E</td>
<td>I</td>
<td>E</td>
</tr>
<tr>
<td>E</td>
<td>E</td>
<td>I</td>
<td>E</td>
</tr>
</tbody>
</table>

Key: I: resolves to internal IP address of head
E: resolves to external IP address of head

DisableLua directive

Syntax: DisableLua = true|false
Default: DisableLua = false

The value of the DisableLua directive determines if Lua code (section L.6) used in monitoring expressions can be executed.

AdvancedConfig directive

Syntax: AdvancedConfig = \{ "<key1>=<value1>", "<key2>=<value2>", ... \}
Default: Commented out in the default cmd.conf file

The AdvancedConfig directive is not part of the standard directives. It takes a set of key/value pairs
as parameters, with each key/value pair allowing a particular functionality, and is quite normal in that respect. However, the functionality of a parameter to this directive is often applicable only under restricted conditions, or for non-standard configurations. The AdvancedConfig parameters are therefore generally not recommended, nor are they documented, for use by the administrator.

Like for the other directives, only one AdvancedConfig directive line is used. This means that whatever functionality is to be enabled by this directive, its corresponding parameters must be added to that one line. These key/value pairs are therefore added by appending them to any existing AdvancedConfig key/value pairs, which means that the directive line can be a long list of key/value pairs to do with a variety of configurations.

GlobalConfig directive

Syntax: GlobalConfig = { "<key1>=<value1>", "<key2>=<value2>", ... }
Default: not in the default cmd.conf file

The GlobalConfig directive is not part of the standard directives. It takes a set of key/value pairs as parameters, with each key/value pair allowing a particular functionality, and is quite normal in that respect. However, the parameter to this directive only needs to be specified on the head node. The non-head node CMDaemons take this value upon connection, which means that the cmd.conf file on the non-head nodes do not need to have this specified. This allows nodes with a boot role (section 5.1.6) to carry out, for example, PXEBootOverHTTP (page 656).

Like for the other directives, only one GlobalConfig directive line is used. This means that whatever functionality is to be enabled by this directive, its corresponding parameters must be added to that one line. These key/value pairs are therefore added by appending them to any existing GlobalConfig key/value pairs, which means that the directive line can be a long list of key/value pairs to do with a variety of configurations.

ScriptEnvironment directive

Syntax: ScriptEnvironment = { "CMD_ENV1=<value1>", "CMD_ENV2=<value2>", ... }
Default: Commented out in the default cmd.conf file

The ScriptEnvironment directive sets extra environment variables for CMDaemon and child processes. For example, if CMDaemon is running behind a web proxy, then the environment variable http_proxy may need to be set for it. If, for example, the proxy is the host brawndo, and it is accessed via port 8080 using a username/password pair of joe/electrolytes, then the directive becomes:

ScriptEnvironment = { "http_proxy=joe:electrolytes@brawndo:8080" }  

BurnSpoolDir directive

Syntax: BurnSpoolDir = path
Default: BurnSpoolDir = "/var/spool/burn/

The BurnSpoolDir directive specifies the directory under which node burn log files are placed (Chapter 8 of the Installation Manual). The log files are logged under a directory named after the booting MAC address of the NIC of the node. For example, for a MAC address of 00:0c:29:92:55:5e the directory is /var/spool/burn/00-0c-29-92-55-5e.

IdleThreshold directive

Syntax: IdleThreshold = number
Default: IdleThreshold = 1.0

© Bright Computing, Inc.
The IdleThreshold directive sets a threshold value for loadone. If loadone exceeds this value, then data producers that have only when idle (page 478) set to true (enabled), will not run. If the data producer is sampled on a regular node rather than on the head node, then cmd.conf on the regular node should be modified and its CMDaemon restarted.

**MonitoringPath directive**

**Syntax:** AdvancedConfig = {"MonitoringPath=path", ...}
**Default:** Implicit value: "MonitoringPath=/var/spool/cmd/monitoring/"

MonitoringPath is a parameter of the AdvancedConfig (page 649) directive. Its value determines the path of the directory in which monitoring data is saved (section 13.8).

**MaxServiceFailureCount directive**

**Syntax:** AdvancedConfig = {"MaxServiceFailureCount=number", ...}
**Default:** Implicit value: "MaxServiceFailureCount=10"

MaxServiceFailureCount is a parameter of the AdvancedConfig (page 649) directive. Its value determines the number of times a service failure event is logged (page 110). Restart attempts on the service still continue when this number is exceeded.

**InitdScriptTimeout directive**

**Syntax:** AdvancedConfig = {"InitdScriptTimeout[.service]=timeout", ...}
**Default:** Implicit value: "InitdScriptTimeout=30"

InitdScriptTimeout is a parameter of the AdvancedConfig (page 649) directive. It can be set globally or locally:

- **Global (all services)**
  InitdScriptTimeout can be set as a global value for init scripts, by assigning timeout as a period in seconds. If an init script fails to start up its service within this period, then CMDaemon kills the service and attempts to restart it.
    - If InitdScriptTimeout has a value for timeout set, then all init scripts have a default timeout of timeout seconds.
    - If InitdScriptTimeout has no timeout value set, then all init scripts have a default timeout of 30 seconds.

- **Local (for a specific service)**
  If InitdScriptTimeout.service is assigned a timeout value, then the init script for that service times out in timeout seconds. This timeout overrides, for that service only, any existing global default timeout.

When a timeout happens for an init script attempting to start a service, the event is logged. If the number of restart attempts exceeds the value determined by the MaxServiceFailureCount directive (page 651), then the event is no longer logged, but the restart attempts continue.

**Example**

An fhgfs startup takes a bit longer than 30 seconds, and therefore times out with the default timeout value of 30s. This results in the following logs in /var/log/cmdaemon:

© Bright Computing, Inc.
Here, service is fhgfs-client, so setting the parameter can be done with:

```
AdvancedConfig = { ..., "initdScriptTimeout.fhgfs-client=60", ...}
```

This allows a more generous timeout of 60 seconds instead.

Restarting CMDaemon then should allow the fhgs startup to complete

```
# service cmd restart
```

A more refined approach that avoids a complete CMDaemon restart would be to execute a reset (page 110) on the fhgfs-client from within CMDaemon, as follows:

```
[bright82->category[default]->services[fhgfs-client]]% reset fhgfs-client
Successfully reset service fhgfs-client on: node001,node002
[bright82->category[default]->services[fhgfs-client]]%
```

**CMDaemonListenOnInterfaces directive**

**Syntax:** `AdvancedConfig = {"CMDaemonListenOnInterfaces=<interfaces>", ...}`

**Default:** all interfaces listening to port 8081

CMDaemonListenOnInterfaces is a parameter of the AdvancedConfig (page 649) directive.

When set explicitly, CMDaemon listens only to the interfaces listed in `<interfaces>`. The form of `<interfaces>` is a comma-separated list of interface device names:

**Example**

```
CMDaemonListenOnInterfaces=eth0,eth1,eth0:0,eth0:1
```

If the interface list item `lo` is omitted from the list of names, it will still be listened to. This is because CMDaemon must always be able to talk to itself on the loopback interface.

**CookieCooldownTime directive**

**Syntax:** `AdvancedConfig = {"CookieCooldownTime=number from 60 to 86400", ...}`

**Default:** 900

CookieCooldownTime is a parameter of the AdvancedConfig (page 649) directive.

It defines the number of seconds until the Bright View connection to CMDaemon times out, if there is no user activity at the Bright View client side.

**OpenStackVXLANGroup directive**

**Syntax:** `AdvancedConfig = {"OpenStackVXLANGroup=IP address", ...}`

**Default:** 224.0.0.1

OpenStackVXLANGroup is a parameter of the AdvancedConfig (page 649) directive.

It defines the multicast address used by OpenStack for broadcast traffic with VXLAN networking (page 30 of the OpenStack Deployment Manual). If an application already uses the default IP address, then another IP address can be specified for VXLAN networking needs with this directive.
MaxSGEJobLimit

Syntax: MaxSGEJobLimit = number
Default: Implicit value: MaxSGEJobLimit = 0

This sets the maximum number of lines that is displayed in the jobs window. By default it is set to 0, which means there is no limit set.

However, if the number of jobs is very large, then parsing them takes time, and can result in a high load on CMDaemon. Setting a limit can help CMDaemon cope with such a case.

RsyncHardLinks directive

Syntax: AdvancedConfig = {"RsyncHardLinks=0|1", ...}
Default: 1

RsyncHardLinks is a parameter of the AdvancedConfig (page 649) directive. It controls, for the rsync command, the hard links preservation option: -H|--hard-links

If RsyncHardLinks is set to:

• 1: then rsync copies hard links
• 0: then rsync does not copy hard links

RsyncXattrs directive

Syntax: AdvancedConfig = {"RsyncXattrs=-1|0|1", ...}
Default: -1

RsyncXattrs is a parameter of the AdvancedConfig (page 649) directive. It controls, for the rsync command, the extended attributes command line option: -X|--xattrs

If RsyncXattrs is set to:

• 1: then rsync copies extended attributes
• 0: then rsync does not copy extended attributes
• -1: then rsync follows the default setting for whether or not to copy extended attributes.

The default setting copies extended attributes only if the distribution version:

- is RHEL7 (and derivatives) and above,
- or SLES12 and above

RsyncAcls directive

Syntax: AdvancedConfig = {"RsyncAcls=-1|0|1", ...}
Default: -1

RsyncAcls is a parameter of the AdvancedConfig (page 649) directive. It controls, for the rsync command, the access control list (ACL) command line option: -A|--acls

If RsyncAcls is set to:

• 1: then rsync copies ACLs
• 0: then rsync does not copy ACLs
-1: then rsync follows the default setting for whether or not to copy ACLs.

The default setting copies ACLs only if the distribution version:

- is RHEL7 (and derivatives), and above,
- or SLES12 and above

**<wlm> Timeout directives:**

*CondorTimeout directive*

*PBSProTimeout directive*

*SGETimeout directive*

*SlurmTimeout directive*

*LSFTimeout directive*

**Syntax:**

```
AdvancedConfig = {"<wlm>Timeout=<number>", ...}
```

where: `<wlm>` is one of:

- Condor
- PBSPro
- SGE
- Slurm
- LSF

**Default:** 120

<wlm> Timeout is a parameter of the AdvancedConfig (page 649) directive. It is the maximum time, in seconds, that CMDaemon will wait for a response from the job scheduler of the workload manager. Increasing this value is sometimes needed for clusters with demanding job loads. Workload managers for which this value can be set are:

- Condor
- PBSPro
- SGE
- Slurm
- LSF

**SlurmDisableAccountingParsing directive**

**Syntax:**

```
AdvancedConfig = {"SlurmDisableAccountingParsing=0\|1", ...}
```

**Default:** 0

SlurmDisableAccountingParsing is a parameter of the AdvancedConfig (page 649) directive. If set to 1, it disables collection of accounting information for the Slurm workload manager.

**JobsSamplingMetricsInterval directive**

**Syntax:**

```
AdvancedConfig = {"JobsSamplingMetricsInterval=<number>", ...}
```

**Default:** 60

JobsSamplingMetricsInterval is a parameter of the AdvancedConfig (page 649) directive. Its value is a time period, in seconds, and it applies only to metrics associated with job queues. Such metric sampling is carried out with this time period if job queues are added, or if job queues are re-created after disappearing.
**MembershipQueryInterval directive**

Syntax: AdvancedConfig = {"MembershipQueryInterval=<number>", ...}

Default: 4

MembershipQueryInterval is a parameter of the AdvancedConfig (page 649) directive. Its value is a time period, in seconds. This time period value elapses between the checks that CMDaemon makes to determine the node states (section 5.5) in a cluster. If the network is very congested, then a larger value can be used to reduce the network load caused by these checks.

**AddUserScript directive**

Syntax: AdvancedConfig = {"AddUserScript=<path>", ...}

Default: none

AddUserScript is a parameter of the AdvancedConfig (page 649) directive. If this parameter is set to a path leading to a script, and if a new user is added using cmsh or Bright View, then the script is automatically run by CMDaemon, with the username of the new user automatically passed as the first argument to the script. The script has a default timeout of 5 seconds.

**AddUserScriptPasswordInEnvironment directive**

Syntax: AdvancedConfig = {"AddUserScriptPasswordInEnvironment=0|1", ...}

Default: 0

AddUserScriptPasswordInEnvironment is a parameter of the AdvancedConfig (page 649) directive. If this parameter is set to 1, then CMDaemon passes the CMD_USER_PASSWORD environment variable to the script defined by the AddUserScript directive.

**RemoveUserScript directive**

Syntax: AdvancedConfig = {"RemoveUserScript=<path>", ...}

Default: none

RemoveUserScript is a parameter of the AdvancedConfig (page 649) directive. If this parameter is set to a path leading to a script, and if an existing user is removed using cmsh or Bright View, then the script is automatically run by CMDaemon. The script has a default timeout of 5 seconds.

**AddUserScriptTimeout directive**

Syntax: AdvancedConfig = {"AddUserScriptTimeout=<number>", ...}

Default: 5

AddUserScriptTimeout is a parameter of the AdvancedConfig (page 649) directive. It sets the timeout value in seconds, for the script set by AddUserScript.

**RemoveUserScriptTimeout directive**

Syntax: AdvancedConfig = {"RemoveUserScriptTimeout=<number>", ...}

Default: 5

RemoveUserScriptTimeout is a parameter of the AdvancedConfig (page 649) directive. It sets the timeout value in seconds, for the script set by RemoveUserScript.
**AdditionalSubmitHosts directive**

**AdditionalExecHosts directive**

Syntax:

```
AdvancedConfig = {"AdditionalSubmitHosts=<host1>,<host2>,..., ...}
```

or

```
AdvancedConfig = {"AdditionalExecHosts=<host1>,<host2>,..., ...}
```

Default: *none*

The `AdditionalSubmitHosts` and `AdditionalExecHosts` directives are parameters of the `AdvancedConfig` (page 649) directive.

These directives can be used to make Bright Cluster Manager aware of the existence of a submit host or execution host that is outside of Bright Cluster Manager control. The directives can be used with the SGE and UGE workload managers only, so that CMDaemon does not remove such hosts from the workload manager configuration file during a configuration file maintenance run. An example of their use is given on pages 257, and 261.

**PXEBootOverHTTP directive**

Syntax: `GlobalConfig = {"PXEBootOverHTTP=1|0", ...}

Syntax: `AdvancedConfig = {"PXEBootOverHTTP=1|0", ...}

Default: 1

The `PXEBootOverHTTP` directive is a parameter of the `GlobalConfig` (page 650) directive. It is also a parameter of the `AdvancedConfig` (page 649) directive.

The `AdvancedConfig` directive is set on the `cmd.conf` per node image to apply it to the associated nodes. The `GlobalConfig` directive is set on the `cmd.conf` file of the head node(s), which applies the value to all nodes.

If the directive is set to a value of 0, then the provisioner sends the initial boot loader, kernel, and ramdisk via TFTP, to the provisioning node.

If the directive is set to a value of 1, which is the default value, then

- The initial boot loader is still loaded up via TFTP, since very few devices support boot loading over HTTP. However, the kernel and ramdisk are now loaded up via HTTP, which is faster and more reliable.
- The administrator must manually set the value of `SSLPortOnly` (page 636) to no, if needed.
- CMDaemon manages the configuration of `/etc/dhcpd.conf` and `/etc/dhcpd.<network name>.conf` on the head node, unless these files are frozen (page 644).
- CMDaemon provides the HTTP server on port 8080, accessible via `http://master:8080/tftpboot`.

**AutomaticMountAll directive**

Syntax: `AutomaticMountAll=0|1`

Default: 1

If the `AutomaticMountAll` directive is set to the default of 1, then a `mount -a` operation is carried out when a mount change is carried out by CMDaemon.

The `mount -a` operation is to do with attempting to mount devices listed in `/etc/fstab`. It should not be confused with auto-mounting of filesystems, which is to do with mounting an arbitrary device to a filesystem automatically.

© Bright Computing, Inc.
If the `AutomaticMountAll` directive is set to 0, then `/etc/fstab` is written, but the mount `-a` command is not run by CMDaemon. However, the administrator should be aware that since `mount -a` is run by the distribution during booting, a node reboot implements the mount change.

**AllowImageUpdateWithAutoMount directive**

**Syntax:** `AdvancedConfig = {"AllowImageUpdateWithAutoMount=0|1|2|3", ...}

**Default:** 0

The `AllowImageUpdateWithAutoMount` directive is a parameter of the `AdvancedConfig` (page 649) directive. The values it takes decide how an auto-mounted filesystem should be dealt with during image updates (section 5.6.2) or grab image (section 11.5.2). It must be set per software image or per node.

<table>
<thead>
<tr>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>If auto-mount is running, abort provisioning (default)</td>
</tr>
<tr>
<td>1</td>
<td>If auto-mount is running, warn but continue</td>
</tr>
<tr>
<td>2</td>
<td>Do not check auto-mount status. This saves a little time, but it risks data loss, unless the automounted filesystem has been added to <code>excludelistupdate</code>. If the automounted filesystem is not added to <code>excludelistupdate</code>, then if the automounted filesystem happens to be unavailable at the time that an <code>imageupdate</code> is issued, then the rsync process can end up deleting the automounted filesystem contents during the rsync, because it assumes that content should not be there.</td>
</tr>
<tr>
<td>3</td>
<td>Pretend auto-mount is running. This prevents an image update</td>
</tr>
</tbody>
</table>

How an auto-mounted filesystem can be configured using the autofs service in Bright Cluster Manager is discussed in section 3.10. The need for adding the automounted filesystem to `excludelistupdate` is discussed on page 198.

**DNS::options_allow-query directive**

**Syntax:** `AdvancedConfig = {"DNS::options_allow-query=<subnet1>, <subnet2>, ...", ...

**Default:** unset

The `DNS::options_allow-query` directive is a parameter of the `AdvancedConfig` (page 649) directive. If a subnet value is specified, in CIDR notation, then that subnet can query the DNS running on the head node. Setting a subnet places an entry within the `allow-query` section of `/etc/named.conf`. The standard directive `PublicDNS` (page 642) simply adds the entry `0.0.0.0/0` to the `allow-query` section and can be used if no specific subnet needs to be added.

**AllowNullCipherNetwork directive**

**Syntax:** `AdvancedConfig = {"AllowNullCipherNetwork=0|1", ...}

**Default:** AllowNullCipherNetwork=1

The `AllowNullCipherNetwork` directive is a parameter of the `AdvancedConfig` (page 649) directive. If it uses the default value of 1, which is recommended, then communication on internal networks is unencrypted. Changing the value to 0 sets encryption on for the internal network, and should only be done for unusual cases.
**CipherList directive**

**Syntax:** AdvancedConfig = {"CipherList=<ciphers>", ...}

**Default:** CipherList=ALL:!aNULL:!eNULL

The CipherList directive is a parameter of the AdvancedConfig (page 649) directive. It sets the cipher list of the OpenSSL suite that CMDaemon negotiates with clients. Ciphers in the cipher list can be viewed with:

**Example**

```
[root@bright82 ~]# openssl ciphers -v
ECDHE-RSA-AES256-GCM-SHA384 TLSv1.2 Kx=ECDH  Au=RSA  Enc=AESGCM(256)  Mac=AEAD
ECDHE-ECDSA-AES256-GCM-SHA384 TLSv1.2 Kx=ECDH  Au=ECDSA  Enc=AESGCM(256)  Mac=AEAD
ECDHE-RSA-AES256-SHA384 TLSv1.2 Kx=ECDH  Au=RSA  Enc=AES(256)  Mac=SHA384
ECDHE-ECDSA-AES256-SHA384 TLSv1.2 Kx=ECDH  Au=ECDSA  Enc=AES(256)  Mac=SHA384
ECDHE-RSA-AES256-SHA SSLv3 Kx=ECDH  Au=RSA  Enc=AES(256)  Mac=SHA1
```

The columns are: the cipher name, SSL protocol, key exchange used, authentication mechanism, encryption used, and MAC digest algorithm.

Ciphers can be specified for the CipherList directive according to the specification described in man (1) ciphers. For example, as:

**Example**

```
AdvancedConfig = {"CipherList=ALL:!aNULL:!ADH:!eNULL:!LOW:!EXP:RC4+RSA:+HIGH:+MEDIUM"}
```

An analogous non-CMDaemon directive is found in LDAP client-server negotiation, where TLS Cipher Suite can be set in the slapd.conf file, described in man(5) slapd.conf.

**SSLServerMethod directive (with TLS masking)**

**Syntax:** AdvancedConfig = {"SSLServerMethod=TLS <versionnumber>", ...}

**Masking syntax:** AdvancedConfig = {"TLS <versionnumber>=<boolean>", ...}

**Default:** none

The SSLServerMethod directive is a parameter of the AdvancedConfig (page 649) directive. It sets the SSL server method for the OpenSSL suite that CMDaemon negotiates with clients. Possible values for <versionnumber> are:

- 1
- 1.1
- 1.2
- 1.3

By default, with no SSLServerMethod directive, all TLS versions are supported. If the SSLServerMethod directive is specified, then only that TLS version is negotiated. For example, to allow only TLS 1.2 negotiation:

**Example**

```
AdvancedConfig = { "SSLServerMethod=TLS 1.2", ... }
```
The TLS masking syntax allows each method to be set concurrently. For example, TLS 1 and TLS 1.1 could be disabled, and TLS 1.2 and TLS 1.3 could be enabled, with:

Example

AdvancedConfig = { "TLS 1=0", "TLS 1.1=0", "TLS 1.2=1", "TLS 1.3=1", ... }

**JobInformationDisabled directive**

Syntax: AdvancedConfig = {"JobInformationDisabled=0|1", ...}

Default: JobInformationDisabled=0

The JobInformationDisabled directive is a parameter of the AdvancedConfig (page 649) directive. If set to 1 it fully disables job-based monitoring (section 12.8).

For a cluster that is running millions of jobs at a time, job-based monitoring can typically consume significant resources. The monitoring of data for so many small jobs is typically not useful and not of interest. For such a case, setting this directive improves cluster performance.

**JobInformationKeepDuration directive**

Syntax: AdvancedConfig = {"JobInformationKeepDuration=<number>", ...}

Default: JobInformationKeepDuration=2419200

The JobInformationKeepDuration directive is a parameter of the AdvancedConfig (page 649) directive. It takes on a value in seconds. If a job has finished more than that many seconds ago, then it will be removed along with all its monitoring data from the database. By default it is set to 28 days (24 × 3600 × 28 seconds).

**CacheJobsTime directive**

Syntax: AdvancedConfig = {"CacheJobsTime=<number>", ...}

Default: CacheJobsTime=3600

The CacheJobsTime directive is a parameter of the AdvancedConfig (page 649) directive. It takes on a value in seconds. If a job has finished more than that many seconds ago, then it will be removed along with all its monitoring data from the memory. By default it is set to 1 hour.

**JobInformationKeepCount directive**

Syntax: AdvancedConfig = {"JobInformationKeepCount=<number>", ...}

Default: JobInformationKeepCount=8192

The JobInformationKeepCount directive is a parameter of the AdvancedConfig (page 649) directive. If the total number of jobs is greater than (JobInformationKeepCount + 10%), then the job record and its data content are discarded, starting from the oldest job first, until the total number of jobs remaining becomes JobInformationKeepCount. If it is set to 0, then none of the job records and content are removed.

The maximum value for this directive is 100 million.

**JobInformationMinimalJobDuration directive**

Syntax: AdvancedConfig = {"JobInformationMinimalJobDuration=<number>", ...}

Default: JobInformationMinimalJobDuration=0

© Bright Computing, Inc.
The JobInformationMinimalJobDuration directive is a parameter of the AdvancedConfig (page 649) directive. If set, then jobs that run for less than this number of seconds are not stored in the cache. Its default value of 0 seconds means that all jobs are handled.

**JobInformationFlushInterval directive**

**Syntax:** `AdvancedConfig = {"JobInformationFlushInterval=<number>", ...}`  
**Default:** `JobInformationFlushInterval=600`

The JobInformationFlushInterval directive is a parameter of the AdvancedConfig (page 649) directive. If this interval, in seconds, is set, then the cache is flushed to the database with that interval. Its default value is 10 minutes (10 × 60 seconds). Values of around 30 seconds or less will conflict with the default CMDaemon maintenance timeout value of 30 seconds, and will mostly simply add load.

**ActionLoggerFile directive**

**Syntax:** `AdvancedConfig = {"ActionLoggerFile=filename", ...}`  
**Default:** `/var/spool/cmd/actions.log`

The directive is a parameter of the AdvancedConfig (page 649) directive. Its value overrides the default path.

The directive needs to be implemented per node or image.

**ActionLoggerOnSuccess directive**

**Syntax:** `AdvancedConfig = {"ActionLoggerOnSuccess=0|1", ...}`  
**Default:** `ActionLoggerOnSuccess=0`

The directive is a parameter of the AdvancedConfig (page 649) directive. By default, only failed actions are logged. Successful actions are also logged if setting `ActionLoggerOnSuccess=1`

The directive needs to be implemented per node or image.

The log file shows timestamped output with one line per run for an action script, with the script response.

**Example**

```
(time) /path/to/action/script [ timeout ]
(time) /path/to/action/script [ failed ] (exit code: 1)
(time) /path/to/action/script [ success ]
```

The success line only appears if `ActionLoggerOnSuccess=1`.

**FailoverPowerRetries directive**

**Syntax:** `AdvancedConfig = {"FailoverPowerRetries=<number>", ...}`  
**Default:** `FailoverPowerRetries=5`

The FailoverPowerRetries directive is a parameter of the AdvancedConfig (page 649) directive.

After a decision to carry out the failover has been made, CMDaemon sends a power off command to the BMC of the head node that is meant to be powered off. If the power off command fails, then on getting the fail response, CMDaemon waits for 1 second. After that second, it sends out a power off command again. The value of FailoverPowerRetries is the number of times that CMDaemon retries
sending the power off command to the BMC of the head node that is intended to be powered off during the failover, if the response from the power off command remains a fail response.

The power down attempts cease, either when the BMC reports that the head node is OFF, or when the number of attempts reaches the FailoverPowerRetries value.

FailoverPowerRetries takes a maximum value of 120.

A value of 0 means that 1 attempt to power off is carried out during failover, but no retry is attempted after the first attempt.

Because CMDaemon waits for a period of 1s before checking for an OFF response, it means that the number of retries is about the same as the number of seconds before CMDaemon decides that powering off has failed, unless the OFF response also takes some time to get to CMDaemon.

Increasing the value for this directive can be useful for some BMC cards that take longer than about 5s to report their power status, because the power off attempt may otherwise time out.
Disk Partitioning And RAID Configuration

Disk partitioning is initially configured on the head node and regular nodes during installation (section 3.3.17 of the Installation Manual).

For the head node it cannot be changed from within the Bright Cluster Manager after implementation, and the head node is therefore not considered further in this section.

For regular nodes, partitioning can be changed after the initial configuration, by specifying a particular set of values according to the XML partitioning schema described in section D.1.

For example, for regular nodes, changing the value set for the XML tag of:

```xml
<xsl:element name='filesystem'>
```

decides which filesystem type out of ext2, ext3, ext4, or xfs is used. The changes are implemented during the node partitioning stage of the node-installer (section 5.4.6).

Diskless operation can also be implemented by using an appropriate XML file. This is introduced in section 3.9.1.

Software or hardware RAID configuration can also be set for the regular nodes. The layouts must be specified following the XML schema files stored on the head node in the directory `/cm/node-installer/scripts/`:

- Software RAID configuration is set in the global partitioning XML schema file `disks.xsd` (section D.1).
- Hardware RAID configuration is set in the separate hardware RAID XML schema file `raid.xsd` (section D.2).

D.1 Structure Of Partitioning Definition—The Global Partitioning XML Schema File

In Bright Cluster Manager, regular node partitioning setups have their global structure defined using an XML schema, which is installed on the head node in `/cm/node-installer/scripts/disks.xsd`. This schema does not include a hardware RAID description. The hardware RAID schema is defined separately in the file `raid.xsd` (section D.2).

Examples of schemas in use, with and without hardware RAID, are given in sections D.3 and beyond. An XML file can be validated against a schema using the `xmllint` tool:

Example

```bash
[root@bright82 ~]# xmllint --noout --schema /cm/node-installer/scripts/disks.xsd \\ /cm/local/apps/cmd/etc/htdocs/disk-setup/slave-diskless.xml \\
/cm/local/apps/cmd/etc/htdocs/disk-setup/slave-diskless.xml validates
[root@bright82 ~]#
```

© Bright Computing, Inc.
The disks.xsd schema listed next also uses \ as a continuation mark on the printed page, to indicate the line continues. That is not valid XML. The \ marks, at end of the lines associated with the stripeSize attribute, must therefore be removed and those lines joined up into one big line, to make the XML file machine-parseable.

**XML schema for partitioning**

```xml
<?xml version='1.0'?>

<!--
# Copyright (c) 2004-2017 Bright Computing, Inc. All Rights Reserved.
#
# This software is the confidential and proprietary information of
# Bright Computing, Inc. ("Confidential Information"). You shall not
# disclose such Confidential Information and shall use it only in
# accordance with the terms of the license agreement you entered into
# with Bright Computing, Inc.
-->

<xs:schema xmlns:xs='http://www.w3.org/2001/XMLSchema' elementFormDefault='qualified'>

<xs:element name='diskSetup'>
  <xs:complexType>
    <xs:sequence>
      <xs:element name='diskless' type='diskless' minOccurs='0' maxOccurs='1'/>
      <xs:element name='device' type='device' minOccurs='0' maxOccurs='unbounded'/>
      <xs:element name='raid' type='raid' minOccurs='0' maxOccurs='unbounded'/>
      <xs:element name='volumeGroup' type='volumeGroup' minOccurs='0' maxOccurs='unbounded'/>
      <xs:element name='subVolumes' type='subVolumes' minOccurs='0' maxOccurs='unbounded'/>
    </xs:sequence>
  </xs:complexType>
</xs:element>

<x:s:complexType>
  <xs:sequence>
    <xs:element name='diskless' type='diskless' minOccurs='0' maxOccurs='1'/>
    <xs:element name='device' type='device' minOccurs='0' maxOccurs='unbounded'/>
    <xs:element name='raid' type='raid' minOccurs='0' maxOccurs='unbounded'/>
    <xs:element name='volumeGroup' type='volumeGroup' minOccurs='0' maxOccurs='unbounded'/>
    <xs:element name='subVolumes' type='subVolumes' minOccurs='0' maxOccurs='unbounded'/>
  </xs:sequence>
</xs:complexType>

<x:s:key name='partitionAndRaidIds'>
  <xs:selector xpath='//raid|//partition'/>
  <xs:field xpath='@id'/>
</xs:key>
```

© Bright Computing, Inc.
<xs:keyref name='raidMemberIds' refer='partitionAndRaidIds'>
  <xs:selector xpath='.//raid/member' />
  <xs:field xpath='.' />
</xs:keyref>

<xs:keyref name='volumeGroupPhysicalVolumes' refer='partitionAndRaidIds'>
  <xs:selector xpath='.//volumeGroup/physicalVolumes/member' />
  <xs:field xpath='.' />
</xs:keyref>

<xs:keyref name='subVolumeIds' refer='partitionAndRaidIds'>
  <xs:selector xpath='.//subVolumes' />
  <xs:field xpath='@parent' />
</xs:keyref>

<xs:unique name='raidAndVolumeMembersUnique'>
  <xs:selector xpath='.' />
  <xs:field xpath='.' />
</xs:unique>

<xs:unique name='deviceNodesUnique'>
  <xs:selector xpath='.//device/blockdev' />
  <xs:field xpath='.' />
  <xs:field xpath='@mode' />
</xs:unique>

<xs:unique name='mountPointsUnique'>
  <xs:selector xpath='.' />
  <xs:field xpath='.' />
</xs:unique>

<xs:unique name='assertNamesUnique'>
  <xs:selector xpath='.' />
  <xs:field xpath='@name' />
</xs:unique>

</xs:element>

<xs:complexType name='diskless'>
  <xs:attribute name='maxMemSize' type='memSize' use='required' />
  <xs:attribute name='mountOptions' type='xs:string' />
</xs:complexType>

<xs:simpleType name='memSize'>
  <xs:restriction base='xs:string'>
    <xs:pattern value='([0-9][MG])|100%|[0-9-][0-9]%|0'/>
  </xs:restriction>
</xs:simpleType>

<xs:simpleType name='stripeSize'>
  <xs:restriction base='xs:string'>
    <xs:pattern value='4|8|16|32|64|128|256|512|1024|1K|2048|2K|4096|4K|8192|8K|16384|16K|32768|32K|65536|64K|131072|128K|262144|256K|524288|512K|1048576|1024K|1M'/> 
  </xs:restriction>
</xs:simpleType>
<xs:simpleType>
  <xs:restriction base="xs:string">
    <xs:pattern value="gpt|msdos"/>
  </xs:restriction>
</xs:simpleType>

<xs:element name='alignMiB' type='xs:boolean' minOccurs='0' maxOccurs='1'/>
<xs:element name='partitionTable' minOccurs='0' maxOccurs='1'>
  <xs:simpleType>
    <xs:restriction base="xs:string">
      <xs:pattern value="gpt|msdos"/>
    </xs:restriction>
  </xs:simpleType>
</xs:element>
<xs:element name='partition' type='partition' minOccurs='1' maxOccurs='unbounded'/>
<xs:sequence>
  <xs:element name='cephosdassociation' type='xs:string' minOccurs='0' maxOccurs='1'/>
  <xs:element name='size' type='relativeOrAbsoluteSize'/>
  <xs:element name='type'>
    <xs:simpleType>
      <xs:restriction base='xs:string'>
        <xs:enumeration value='linux'/>
        <xs:enumeration value='linux swap'/>
        <xs:enumeration value='linux raid'/>
        <xs:enumeration value='linux lvm'/>
        <xs:enumeration value='unspecified'/>
      </xs:restriction>
    </xs:simpleType>
  </xs:element>
  <xs:element name='priority' minOccurs='0' maxOccurs='1'>
    <xs:simpleType>
      <xs:restriction base="xs:integer">
        <xs:minInclusive value="0"/>
        <xs:maxInclusive value="32767"/>
      </xs:restriction>
    </xs:simpleType>
  </xs:element>
</xs:sequence>
<xs:attribute name='id' type='xs:string' use='required'/>
<xs:attribute name='partitiontype' type='xs:string'/>
</xs:complexType>

<xs:group name='filesystem'>
  <xs:sequence>
    <xs:element name='filesystem'>
      <xs:simpleType>
        <xs:restriction base='xs:string'>
          <xs:pattern value="gpt|msdos"/>
        </xs:restriction>
      </xs:simpleType>
    </xs:element>
  </xs:sequence>
</xs:group>

© Bright Computing, Inc.
Examples Of Element Types In XML Schema

© Bright Computing, Inc.
### D.2 Structure Of Hardware RAID Definition—The Hardware RAID XML Schema File

Hardware RAID can be specified using an XML schema, stored on the head node in `/cm/node-installer/scripts/raid.xsd`. The full schema specification is listed here, while schema examples are listed in section D.4.1.

```xml
<?xml version="1.0" encoding="ISO-8859-1" ?>
<!--
# Copyright (c) 2004-2010 Bright Computing, Inc. All Rights Reserved.
# This software is the confidential and proprietary information of
# Bright Computing, Inc. ("Confidential Information"). You shall not
# disclose such Confidential Information and shall use it only in
# accordance with the terms of the license agreement you entered into
# with Bright Computing, Inc.

This is the XML schema description of the hardware RAID layout XML file. It can be used by software to validate partitioning XML files. There are however a few things the schema does not check:
- All of the spans (drive groups) in an raidArray must have the same number of drives.
- There can only be one volume with a 'max' size on a particular array and this must be the last volume in the array.
- If there is only one enclosure defined for a particular RAID controller the actual enclosureID can be omitted by using 'auto' instead. Otherwise, the actual enclosureID must be specified.
-->

<x:schema xmlns:xs="http://www.w3.org/2001/XMLSchema">

<x:simpleType name="raidLevel">
  <xs:restriction base="xs:nonNegativeInteger">
    <xs:pattern value="0|1|5|10|50"/>
  </xs:restriction>
</x:simpleType>

<x:simpleType name="volumeSize">
  <xs:restriction base="xs:string">
    <xs:pattern value="[0-9]1,5[GT]|max"/>
  </xs:restriction>
</x:simpleType>

<x:simpleType name="stripeSize">
  <xs:restriction base="xs:string">
    <xs:pattern value="[0-9]\,5\,[MGT]\,max"/>
  </xs:restriction>
</x:simpleType>
```

---

**Name Of Type** | **Example Values**
--- | ---
**size** | 10G, 128M, 1T, 2.5T, 1/3, 33.333%, auto, max
**device** | `/dev/sda`, `/dev/hda`, `/dev/cciss/c0d0`
**partition** | linux, linux raid, linux swap, unspecified
**filesystem** | ext2, ext3, ext4, xfs
<xs:simpleType name="cachePolicy">
    <xs:restriction base="xs:string">
        <xs:pattern value="Cached|Direct"/>
    </xs:restriction>
</xs:simpleType>

<xs:simpleType name="readPolicy">
    <xs:restriction base="xs:string">
        <xs:pattern value="NORA|RA|ADRA"/>
    </xs:restriction>
</xs:simpleType>

<xs:simpleType name="writePolicy">
    <xs:restriction base="xs:string">
        <xs:pattern value="WT|WB"/>
    </xs:restriction>
</xs:simpleType>

<xs:simpleType name="enclosureID">
    <xs:restriction base="xs:string">
        <xs:pattern value="auto|[0-9]{1,4}"/>
    </xs:restriction>
</xs:simpleType>

<xs:simpleType name="slotNumber">
    <xs:restriction base="xs:nonNegativeInteger">
        <xs:pattern value="[0-9]{1,2}"/>
    </xs:restriction>
</xs:simpleType>
<xs:complexType>
  <xs:sequence>
    <xs:element name="stripeSize" type="stripeSize"/>
    <xs:element name="cachePolicy" type="cachePolicy"/>
    <xs:element name="readPolicy" type="readPolicy"/>
    <xs:element name="writePolicy" type="writePolicy"/>
    <xs:element name="size" type="volumeSize"/>
  </xs:sequence>
</xs:complexType>

<xs:choice>
  <xs:element name="device" maxOccurs="unbounded">
    <xs:complexType>
      <xs:sequence>
        <xs:element name="enclosureID" type="enclosureID"/>
        <xs:element name="slotNumber" type="slotNumber"/>
      </xs:sequence>
    </xs:complexType>
  </xs:element>

  <xs:element name="span" minOccurs="2" maxOccurs="8">
    <xs:complexType>
      <xs:sequence>
        <xs:element name="device" maxOccurs="unbounded">
          <xs:complexType>
            <xs:sequence>
              <xs:element name="enclosureID" type="enclosureID"/>
              <xs:element name="slotNumber" type="slotNumber"/>
            </xs:sequence>
          </xs:complexType>
        </xs:element>
      </xs:sequence>
    </xs:complexType>
  </xs:element>
</xs:choice>
</xs:complexType>
D.3 Example: Default Node Partitioning

The following example follows the schema specification of section D.1, and shows the default layout used for regular nodes:

Example

```xml
<?xml version="1.0" encoding="UTF-8"?>
<diskSetup xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance">
  <device>
    <blockdev>/dev/sda</blockdev>
    <blockdev>/dev/hda</blockdev>
    <blockdev>/dev/vda</blockdev>
    <blockdev>/dev/xvda</blockdev>
    <blockdev>/dev/cciss/c0d0</blockdev>
    <blockdev mode="cloud">/dev/sdb</blockdev>
    <blockdev mode="cloud">/dev/hdb</blockdev>
    <blockdev mode="cloud">/dev/vdb</blockdev>
    <blockdev mode="cloud">/dev/xvdb</blockdev>
    <blockdev mode="cloud">/dev/xvdf</blockdev>
    <partition id="a1">
      <size>20G</size>
      <type>linux</type>
      <filesystem>ext3</filesystem>
      <mountPoint>/</mountPoint>
      <mountOptions>defaults,noatime,nodiratime</mountOptions>
    </partition>
    <partition id="a2">
      <size>6G</size>
      <type>linux</type>
      <filesystem>ext3</filesystem>
      <mountPoint>/var</mountPoint>
      <mountOptions>defaults,noatime,nodiratime</mountOptions>
    </partition>
    <partition id="a3">
      <size>2G</size>
      <type>linux</type>
      <filesystem>ext3</filesystem>
      <mountPoint>/tmp</mountPoint>
      <mountOptions>defaults,noatime,nodiratime,nosuid,nodev</mountOptions>
    </partition>
    <partition id="a4">
      <size>12G</size>
      <type>linux swap</type>
    </partition>
    <partition id="a5">
      <size>max</size>
  </device>
</diskSetup>
```
The example assumes a single disk. Another disk can be added by adding another pair of 
<device><device> tags and filling in the partitioning specifications for the next disk. Because multiple 
blockdev tags are used, the node-installer first tries to use /dev/sda, then /dev/hda, then /dev/vda (virtio disks), then /dev/xvda (xen disks), and so on. Cloud devices can also be accessed using the 
mode="cloud" option. Removing block devices from the layout if they are not going to be used does no harm.

For each partition, a size tag is specified. Sizes can be specified using megabytes (e.g. 500M), gigabytes (e.g. 50G) or terabytes (e.g. 2T or 4.5T). Relative sizes, without units, can be used in the form of fractions (e.g. 2/3) or percentages (e.g. 70%), which can be useful for disk sizes that are not known in advance.

Small differences in size do not trigger a full install for existing relative partitions.

For swap partitions, a size of auto sets a swap partition to twice the node memory size. If there is 
more than one swap partition, then the priority tag can be set so that the partition with the higher priority is used first.

For a device, the attribute max for a size tag forces the last device in the partition to use all remaining 
space, and if needed, adjusts the implementation of the sequence of size tags in the remaining partitions 
accordingly. The use of max for a partition is convenient.

In the example, all filesystems are specified as ext3. Valid alternatives are ext2, ext4 and xfs.

The mount man page has more details on mount options. If the mountOptions tag is left empty, its value defaults to defaults.

D.4 Example: Hardware RAID Configuration

A prerequisite with hardware RAID is that it must be enabled and configured properly in the BIOS.

If it is enabled and configured correctly, then the hardware RAID configuration can be defined or modified by setting the hardwareraidconfiguration parameter in device or category mode:

Example

[root@bright82 ~]# cmsh
[bright82]% category use default
[bright82->category[default]]% set hardwareraidconfiguration

This opens up an editor in which the XML file can be specified according to the schema in section D.2. XML validation is carried out.

D.4.1 RAID level 0 And RAID 10 Example

In the following configuration the node has two RAID arrays, one in a RAID 0 and the other in a RAID 10 configuration:

- The RAID 0 array contains three volumes and is made up of two hard disks, placed in slots 0 and 1. The volumes have different values for the options and policies.

- The RAID 10 array consists of just one volume and has two spans, in slots 2 and 3. Each span has two hard disks.

Example
Example: Hardware RAID Configuration

```xml
<raidSetup>
  <raidArray>
    <level>0</level>
    <raidVolume>
      <stripeSize>64K</stripeSize>
      <cachePolicy>Direct</cachePolicy>
      <readPolicy>NORA</readPolicy>
      <writePolicy>WT</writePolicy>
      <size>40G</size>
    </raidVolume>
    <raidVolume>
      <stripeSize>128K</stripeSize>
      <cachePolicy>Direct</cachePolicy>
      <readPolicy>RA</readPolicy>
      <writePolicy>WB</writePolicy>
      <size>80G</size>
    </raidVolume>
    <raidVolume>
      <stripeSize>256K</stripeSize>
      <cachePolicy>Cached</cachePolicy>
      <readPolicy>ADRA</readPolicy>
      <writePolicy>WT</writePolicy>
      <size>100G</size>
    </raidVolume>
  </raidArray>
  <raidArray>
    <level>10</level>
    <raidVolume>
      <stripeSize>64K</stripeSize>
      <cachePolicy>Direct</cachePolicy>
      <readPolicy>NORA</readPolicy>
      <writePolicy>WT</writePolicy>
      <size>40G</size>
    </raidVolume>
  </raidArray>
</raidSetup>
```

© Bright Computing, Inc.
D.5 Example: Software RAID

The following example shows a simple software RAID setup:

Example

```xml
<?xml version="1.0" encoding="ISO-8859-1"?>
<diskSetup xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance"
    xsi:noNamespaceSchemaLocation="schema.xsd">
    <device>
        <blockdev>/dev/sda</blockdev>
        <partition id="a1">
            <size>25G</size>
            <type>linux raid</type>
        </partition>
    </device>
    <device>
        <blockdev>/dev/sdb</blockdev>
        <partition id="b1">
            <size>25G</size>
            <type>linux raid</type>
        </partition>
    </device>
    <raid id="r1">
        <member>a1</member>
        <member>b1</member>
        <level>1</level>
        <filesystem>ext3</filesystem>
        <mountPoint>/</mountPoint>
        <mountOptions>defaults,noatime,nodiratime</mountOptions>
    </raid>
</diskSetup>
```
The `level` tag specifies the RAID level used. The following are supported:

- 0 (striping without parity)
- 1 (mirroring)
- 4 (striping with dedicated parity drive)
- 5 (striping with distributed parity)
- 6 (striping with distributed double parity)

The `member` tags must refer to an `id` attribute of a `partition` tag, or an `id` attribute of another `raid` tag. The latter can be used to create, for example, RAID 10 configurations.

The administrator must ensure that the correct RAID kernel module is loaded (section 5.3.2). Including the appropriate module from the following is usually sufficient: `raid0`, `raid1`, `raid4`, `raid5`, `raid6`.

### D.6 Example: Software RAID With Swap

The `<swap>` tag is used to indicate a swap partition in a RAID device specified in the XML schema of section D.1. For example, the following marks a 1GB RAID 1 partition as being used for swap, and the second partition for an ext3 filesystem:

```xml
<?xml version="1.0" encoding="UTF-8"?>
<diskSetup xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance">
  <device>
    <blockdev>/dev/sda</blockdev>
    <partition id="a1">
      <size>1G</size>
      <type>linux raid</type>
    </partition>
    <partition id="a2">
      <size>max</size>
      <type>linux raid</type>
    </partition>
  </device>
  <device>
    <blockdev>/dev/sdb</blockdev>
    <partition id="b1">
      <size>1G</size>
      <type>linux raid</type>
    </partition>
    <partition id="b2">
      <size>max</size>
      <type>linux raid</type>
    </partition>
  </device>
  <raid id="r1">
    <member>a1</member>
    <member>b1</member>
    <level>1</level>
    <swap/>
</diskSetup>
```

© Bright Computing, Inc.
As in section D.5, the appropriate RAID modules must be loaded beforehand.

D.7 Example: Logical Volume Manager

This example shows a simple LVM setup:

Example

```xml
<?xml version="1.0" encoding="UTF-8"?>
<diskSetup xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance">
  <device>
    <blockdev>/dev/sda</blockdev>
    <blockdev>/dev/hda</blockdev>
    <blockdev>/dev/vda</blockdev>
    <blockdev>/dev/xvda</blockdev>
    <blockdev>/dev/cciss/c0d0</blockdev>
    <blockdev mode="cloud">/dev/sdb</blockdev>
    <blockdev mode="cloud">/dev/hdb</blockdev>
    <blockdev mode="cloud">/dev/vdb</blockdev>
    <blockdev mode="cloud">/dev/xvdb</blockdev>
    <blockdev mode="cloud">/dev/xvdf</blockdev>
  </device>
  <partition id="a1">
    <size>512M</size>
    <type>linux</type>
    <filesystem>ext2</filesystem>
    <mountPoint>/boot</mountPoint>
    <mountOptions>defaults,noatime,nodiratime</mountOptions>
  </partition>
  <partition id="a2">
    <size>16G</size>
    <type>linux swap</type>
  </partition>
  <partition id="a3">
    <size>max</size>
    <type>linux lvm</type>
  </partition>
</device>
<volumeGroup>
  <name>vg01</name>
  <extentSize>8M</extentSize>
  <physicalVolumes>
    <member>a3</member>
  </physicalVolumes>
  <logicalVolumes>
    <volume>
```
The member tags must refer to an id attribute of a partition tag, or an id attribute of a raid tag. The administrator must ensure that the dm-mod kernel module is loaded when LVM is used.

D.8  Example: Logical Volume Manager With RAID 1

This example shows an LVM setup, but with the LVM partitions mirrored using RAID 1:

Example

```xml
<?xml version="1.0" encoding="ISO-8859-1"?>
<diskSetup xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance" xsi:noNamespaceSchemaLocation="schema.xsd">
  <device>
    <blockdev>/dev/sda</blockdev>
    <partition id="a1">
      <size>512M</size>
      <type>linux raid</type>
    </partition>
    <partition id="a2">
      <size>max</size>
      <type>linux raid</type>
    </partition>
  </device>
  <device>
    <blockdev>/dev/sdb</blockdev>
    <partition id="b1">
      <size>512M</size>
      <type>linux raid</type>
    </partition>
    <partition id="b2">
      <size>max</size>
      <type>linux raid</type>
    </partition>
  </device>
  <raid id="r1">
    <member>a1</member>
  </raid>
</diskSetup>
```
D.9 Example: Diskless

This example shows a node configured for diskless operation:

Example

<?xml version="1.0" encoding="UTF-8"?>
<diskSetup xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance">

© Bright Computing, Inc.
An example of the implementation of a diskless configuration is given in section 3.9.3.

In diskless mode the software image is transferred by the node-installer to a RAM-based filesystem on the node called tmpfs.

The obvious advantage of running from RAM is the elimination of the physical disk, cutting power consumption and reducing the chance of hardware failure. On the other hand, some of the RAM on the node is then no longer available for user applications.

**Special considerations with diskless mode:**

- **Recommended minimum RAM size:** The available RAM per node should be sufficient to run the OS and the required tasks. At least 4GB is recommended for diskless nodes.

- **The tmpfs size limit:** The maximum amount of RAM that can be used for a filesystem is set with the maxMemSize attribute. A value of 100% allows all of the RAM to be used. The default value is 90%. A value of 0, without the % sign, removes all restrictions.
  
  A limit does not however necessarily prevent the node from crashing, as some processes might not deal properly with a situation when there is no more space left on the filesystem.

- **Persistence issues:** While running as a diskless node, the node is unable to retain any non-shared data each time it reboots. For example the files in /var/log/*, which are normally preserved by the exclude list settings for disked nodes, are lost from RAM during diskless mode reboots. The installmode NOSYNC setting cannot be used with diskless nodes during a node reboot.

- **Leftover disk issues:** Administrators in charge of sensitive environments should be aware that the disk of a node that is now running in diskless mode still contains files from the last time the disk was used, unless the files are explicitly wiped.

- **Reducing the software image size in tmpfs on a diskless node:** To make more RAM available for tasks, the software image size held in RAM can be reduced:
  
  - by removing unnecessary software from the image.
  - by mounting parts of the filesystem in the image over NFS during normal use. This is especially worthwhile for less frequently accessed parts of the image (section 3.10.3).

**D.10 Example: Semi-diskless**

Diskless operation (section D.9) can also be mixed with certain parts of the filesystem on the local physical disk. This frees up RAM which the node can then put to other use. In this example all data in /local is on the physical disk, the rest in RAM.

**Example**

```xml
<?xml version="1.0" encoding="UTF-8"?>

diskSetup xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance"
xsi:noNamespaceSchemaLocation="schema.xsd">
  <diskless maxMemSize="0"></diskless>
  <device>
    <blockdev>/dev/sda</blockdev>
    <partition id="a1">
      <size>max</size>
      <type>linux</type>
  </device>
</diskSetup>
```

© Bright Computing, Inc.
When nodes operate in semi-diskless mode the node-installer always uses `excludelistfullinstall` (section 5.4.7) when synchronizing the software image to memory and disk.

An alternative to using a local disk for freeing up RAM is to use NFS storage, as is described in section 3.10.3.

### D.11 Example: Preventing Accidental Data Loss

Optional tags, `vendor` and `requiredSize`, can be used to prevent accidentally repartitioning the wrong drive. Such a tag use is shown in the following example.

**Example**

```xml
<?xml version="1.0" encoding="ISO-8859-1"?>
<diskSetup xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance"
  xsi:noNamespaceSchemaLocation="schema.xsd">
  <device>
    <blockdev>/dev/sda</blockdev>
    <vendor>Hitachi</vendor>
    <requiredSize>200G</requiredSize>
    <partition id="a1">
      <size>max</size>
      <type>linux</type>
      <filesystem>ext3</filesystem>
      <mountPoint>/</mountPoint>
      <mountOptions>defaults,noatime,nodiratime</mountOptions>
    </partition>
  </device>
  <device>
    <blockdev>/dev/sdb</blockdev>
    <vendor>BigRaid</vendor>
    <requiredSize>2T</requiredSize>
    <partition id="b1">
      <size>max</size>
      <type>linux</type>
      <filesystem>ext3</filesystem>
      <mountPoint>/data</mountPoint>
      <mountOptions>defaults,noatime,nodiratime</mountOptions>
    </partition>
  </device>
</diskSetup>
```

If a vendor or a `requiredSize` element is specified, it is treated as an assertion which is checked by the node-installer. The node-installer reads the drive vendor string from `/sys/block/<drive`
name>/device/vendor. For the assertion to succeed, the ratio of actual disk size to the value specified by requiredSize, should be at least 0.85:1, and at most 1:0.85.

That is: to be able to get past the requiredSize assertion, the actual drive size as seen from fdisk -l should be 85% to about 118% of the asserted size.

If any assertion fails, no partitioning changes will be made to any of the specified devices.

For assertions with drives that are similar or identical in size, and are from the same vendor, the requiredSize and vendor elements are not enough to differentiate between the drives. In such cases, custom assertions (section D.12) can be set for particular drives.

Specifying device assertions is recommended for nodes that contain important data because it protects against a situation where a drive is assigned to an incorrect block device. This can happen, for example, when the first drive, for example /dev/sda, in a multi-drive system is not detected (e.g. due to a hardware failure, or a BIOS update) which could cause the second drive to become known as /dev/sda2, potentially causing much woe.

As an aside, CMDaemon does offer another way, outside of assertions, to avoid wiping out drive data automatically. This is done in cmsh by setting the value of datanode to yes (section 5.4.4).

D.12 Example: Using Custom Assertions

The following example shows the use of the assert tag, which can be added to a device definition:

Example

```xml
<?xml version="1.0" encoding="ISO-8859-1"?>
<diskSetup xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance"
  xsi:noNamespaceSchemaLocation="schema.xsd">
  <device>
    <blockdev>/dev/sda</blockdev>
    <assert name="modelCheck" args="WD800AAJS">
      <![CDATA[
        #!/bin/bash
        if grep -q $1 /sys/block/$ASSERT_DEV/device/model; then
          exit 0
        else
          exit 1
        fi
      ]]>}
    </assert>
    <partition id="a1">
      <size>max</size>
      <type>linux</type>
      <filesystem>ext3</filesystem>
      <mountPoint>/</mountPoint>
      <mountOptions>defaults,noatime,nodiratime</mountOptions>
    </partition>
  </device>
  <device>
    <blockdev>/dev/sdb</blockdev>
    <vendor>BigRaid</vendor>
    <requiredSize>2T</requiredSize>
    <partition id="b1">
      <size>max</size>
    </partition>
  </device>
</diskSetup>
```
The assert tag is similar to the vendor and size tags described in section D.11.

It can be used to define custom assertions. The assertions can be implemented using any script language.

The script can access the environment variables ASSERT_DEV (eg: sda) and ASSERT_NODE (eg: /dev/sda) during the node-installer stage.

Each assert needs to be assigned an arbitrary name and can be passed custom parameters. A non-zero exit code in the assertion causes the node-installer to halt.
Example initialize And finalize Scripts

The node-installer executes any initialize and finalize scripts at particular stages of its 13-step run during node-provisioning (section 5.4). They are sometimes useful for troubleshooting or workarounds during those stages. The scripts are stored in the CMDaemon database, rather than in the filesystem as plain text files, because they run before the node’s init process takes over and establishes the final filesystem.

E.1 When Are They Used?

They are sometimes used as an alternative configuration option out of a choice of other possible options (section 3.15.1). As a solution it can be a bit of a hack, but sometimes there is no reasonable alternative other than using an initialize or finalize script.

An initialize script: is used well before the init process starts, to execute custom commands before partitions and mounting devices are checked. Typically, initialize script commands are related to partitioning, mounting, or initializing special storage hardware. Often an initialize script is needed because the commands in it cannot be stored persistently anywhere else.

A finalize script: (also run before init, but shortly before init starts) is used to set a file configuration or to initialize special hardware, sometimes after a hardware check. It is run in order to make software or hardware work before, or during the later init stage of boot. Thus, often a finalize script is needed because its commands must be executed before init, and the commands cannot be stored persistently anywhere else, or it is needed because a choice between (otherwise non-persistent) configuration files must be made based on the hardware before init starts.

E.2 Accessing From Bright View And cmsh

The initialize and finalize scripts are accessible for viewing and editing:

- In Bright View, via the Node Categories or Nodes window, under the Settings window. The clickpaths for these are:
  - Grouping→Node categories[default]→Edit→Settings
  - Devices→Nodes[node001]→Edit→Settings

- In cmsh, using the category or device modes. The get command is used for viewing the script, and the set command to start up the default text editor to edit the script. Output is truncated in the two following examples at the point where the editor starts up:

© Bright Computing, Inc.
Example

[root@bright82 ~]# cmsh
[bright82]% category use default
[bright82->category[default]]% show | grep script
Parameter Value
-----------------------------------------------
Finalize script <1367 bytes>
Initialize script <0 bytes>
[bright82->category[default]]% set initializescript

Example

[bright82]% device use node001
[bright82->device[node001]]% set finalizescript

E.3 Environment Variables Available To initialize And finalize Scripts

For the initialize and finalize scripts, node-specific customizations can be made from a script using environment variables. The following table shows the available variables with some example values:

<table>
<thead>
<tr>
<th>Variable</th>
<th>Example Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>CMD_ACTIVE_MASTER_IP</td>
<td>10.141.255.254</td>
</tr>
<tr>
<td>CMD_CATEGORY</td>
<td>default</td>
</tr>
<tr>
<td>CMD_CHASSIS</td>
<td>chassis01</td>
</tr>
<tr>
<td>CMD_CHASSIS_IP</td>
<td>10.141.1.1</td>
</tr>
<tr>
<td>CMD_CHASSIS_PASSWORD</td>
<td>ADMIN</td>
</tr>
<tr>
<td>CMD_CHASSIS_SLOT</td>
<td>1</td>
</tr>
<tr>
<td>CMD_CHASSIS_USERNAME</td>
<td>ADMIN</td>
</tr>
<tr>
<td>CMD_CLUSTERNAME</td>
<td>Bright 8.2 Cluster</td>
</tr>
<tr>
<td>CMD_DEVICE_HEIGHT</td>
<td>1</td>
</tr>
<tr>
<td>CMD_DEVICE_POSITION</td>
<td>10</td>
</tr>
<tr>
<td>CMD_DEVICE_TYPE</td>
<td>SlaveNode</td>
</tr>
<tr>
<td>CMD_EthernetSwitch</td>
<td>switch01:1</td>
</tr>
<tr>
<td>CMD_FSEXPORT__SLASH_cm_SLASH_node-installer_ALLOWWRITE</td>
<td>no</td>
</tr>
<tr>
<td>CMD_FSEXPORT__SLASH_cm_SLASH_node-installer_HOSTS</td>
<td>10.141.0.0/16</td>
</tr>
<tr>
<td>CMD_FSEXPORT__SLASH_cm_SLASH_node-installer_PATH</td>
<td>/cm/node-installer</td>
</tr>
<tr>
<td>CMD_FSEXPORTS</td>
<td>_SLASH_cm_SLASH_node-installer</td>
</tr>
<tr>
<td>CMD_FSMOUNT__SLASH_cm_SLASH_shared_DEVICE</td>
<td>master:/cm/shared</td>
</tr>
<tr>
<td>CMD_FSMOUNT__SLASH_cm_SLASH_shared_FILESYSTEM</td>
<td>nfs</td>
</tr>
</tbody>
</table>

...continues
### Table E: Environment Variables For The `initialize` And Finalize Scripts...continued

<table>
<thead>
<tr>
<th>Variable</th>
<th>Example Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>CMD_FSMOUNT_cm_SLASH_shared_MOUNTPOINT</td>
<td>/cm/shared</td>
</tr>
<tr>
<td>CMD_FSMOUNT_cm_SLASH_shared_OPTIONS</td>
<td>rsize=32768,wsize=32768,hard,intr,async</td>
</tr>
<tr>
<td>CMD_FSMOUNT_dev_SLASH_pts_DEVICE</td>
<td>none</td>
</tr>
<tr>
<td>CMD_FSMOUNT_dev_SLASH_pts_FILESYSTEM</td>
<td>devpts</td>
</tr>
<tr>
<td>CMD_FSMOUNT_dev_SLASH_pts_MOUNTPOINT</td>
<td>/dev/pts</td>
</tr>
<tr>
<td>CMD_FSMOUNT_dev_SLASH_pts_OPTIONS</td>
<td>gid=5,mode=620</td>
</tr>
<tr>
<td>CMD_FSMOUNT_dev_SLASH_shm_DEVICE</td>
<td>none</td>
</tr>
<tr>
<td>CMD_FSMOUNT_dev_SLASH_shm_FILESYSTEM</td>
<td>tmpfs</td>
</tr>
<tr>
<td>CMD_FSMOUNT_dev_SLASH_shm_MOUNTPOINT</td>
<td>/dev/shm</td>
</tr>
<tr>
<td>CMD_FSMOUNT_dev_SLASH_shm_OPTIONS</td>
<td>defaults</td>
</tr>
<tr>
<td>CMD_FSMOUNT_home_DEVICE</td>
<td>master:/home</td>
</tr>
<tr>
<td>CMD_FSMOUNT_home_FILESYSTEM</td>
<td>nfs</td>
</tr>
<tr>
<td>CMD_FSMOUNT_home_MOUNTPOINT</td>
<td>home</td>
</tr>
<tr>
<td>CMD_FSMOUNT_home_OPTIONS</td>
<td>rsize=32768,wsize=32768,hard,intr,async</td>
</tr>
<tr>
<td>CMD_FSMOUNT_proc_DEVICE</td>
<td>none</td>
</tr>
<tr>
<td>CMD_FSMOUNT_proc_FILESYSTEM</td>
<td>proc</td>
</tr>
<tr>
<td>CMD_FSMOUNT_proc_MOUNTPOINT</td>
<td>/proc</td>
</tr>
<tr>
<td>CMD_FSMOUNT_proc_OPTIONS</td>
<td>defaults,nosuid</td>
</tr>
<tr>
<td>CMD_FSMOUNT_sys_DEVICE</td>
<td>none</td>
</tr>
<tr>
<td>CMD_FSMOUNT_sys_FILESYSTEM</td>
<td>sysfs</td>
</tr>
<tr>
<td>CMD_FSMOUNT_sys_MOUNTPOINT</td>
<td>/sys</td>
</tr>
<tr>
<td>CMD_FSMOUNT_sys_OPTIONS</td>
<td>defaults</td>
</tr>
<tr>
<td>CMD_FSMOUNTS * _SLASH_dev_SLASH_pts _SLASH_proc _SLASH_sys _SLASH_dev_SLASH_shm _SLASH_cm_SLASH_shared _SLASH_home</td>
<td></td>
</tr>
<tr>
<td>CMD_GATEWAY</td>
<td>10.141.255.254</td>
</tr>
<tr>
<td>CMD_HOSTNAME</td>
<td>node001</td>
</tr>
<tr>
<td>CMD_INSTALLMODE</td>
<td>AUTO</td>
</tr>
<tr>
<td>CMD_INTERFACE_eth0_IP **</td>
<td>10.141.0.1</td>
</tr>
<tr>
<td>CMD_INTERFACE_eth0_MTU **</td>
<td>1500</td>
</tr>
<tr>
<td>CMD_INTERFACE_eth0_NETMASK **</td>
<td>255.255.0.0</td>
</tr>
<tr>
<td>CMD_INTERFACE_eth0_TYPE **</td>
<td>physical</td>
</tr>
<tr>
<td>CMD_INTERFACES *</td>
<td>eth0 eth1 eth2 ipmi0</td>
</tr>
<tr>
<td>CMD_IP</td>
<td>10.141.0.1</td>
</tr>
<tr>
<td>CMD_MAC</td>
<td>00:00:00:00:00:01</td>
</tr>
<tr>
<td>CMD_PARTITION</td>
<td>base</td>
</tr>
</tbody>
</table>

...continues
Table E: Environment Variables For The initialize And Finalize Scripts...continued

<table>
<thead>
<tr>
<th>Variable</th>
<th>Example Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>CMD_PASSIVE_MASTER_IP</td>
<td>10.141.255.253</td>
</tr>
<tr>
<td>CMD_PDUS</td>
<td></td>
</tr>
<tr>
<td>CMDPOWER_CONTROL</td>
<td>custom</td>
</tr>
<tr>
<td>CMD_RACK</td>
<td>rack01</td>
</tr>
<tr>
<td>CMD_RACK_HEIGHT</td>
<td>42</td>
</tr>
<tr>
<td>CMD_RACK_ROOM</td>
<td>serverroom</td>
</tr>
<tr>
<td>CMD_ROLES</td>
<td>sgeclient storage</td>
</tr>
<tr>
<td>CMD_SHARED_MASTER_IP</td>
<td>10.141.255.252</td>
</tr>
<tr>
<td>CMD_SOFTWAREIMAGE_PATH</td>
<td>/cm/images/default-image</td>
</tr>
<tr>
<td>CMD_SOFTWAREIMAGE</td>
<td>default-image</td>
</tr>
<tr>
<td>CMD_TAG</td>
<td>00000000a000</td>
</tr>
<tr>
<td>CMD_USERDEFINED1</td>
<td>var1</td>
</tr>
<tr>
<td>CMD_USERDEFINED2</td>
<td>var2</td>
</tr>
</tbody>
</table>

* The value for this variable is a string with spaces, not an array. Eg:
  CMD_FSMOUNTS="_SLASH_dev_SLASH_pts _SLASH_proc _SLASH_sys _SLASH_dev_SLASH_shm ..."

** The name of this variable varies according to the interfaces available. So,
eth0 can be replaced by eth1, eth2, ipmi0, and so on.

E.4 Using Environment Variables Stored In Multiple Variables

Some data values, such as those related to interfaces (CMD_INTERFACES_*), mount points
(CMD_FSMOUNT_*SLASH_*) and exports (CMD_FSEXPORT_*SLASH_cm_*SLASH_node-installer_*) are
stored in multiple variables. The code example below shows how they can be used:

Example

#!/bin/bash

for interface in $CMD_INTERFACES
do
  eval type=\$CMD_INTERFACE_${interface}_TYPE
  eval ip=\$CMD_INTERFACE_${interface}_IP
  eval mask=\$CMD_INTERFACE_${interface}_NETMASK

  echo "interface type=type"
  echo "interface ip=ip"
  echo "interface netmask=mask"
done

For remotely mounted devices, the name of the environment variables for mount entries have the
following naming convention:
E.5 Storing A Configuration To A Filesystem

### E.5.1 Storing With Initialize Scripts

The initialize script (section 5.4.5) runs after the install-mode type and execution have been determined (section 5.4.4), but before unloading specific drivers and before partitions are checked and filesystems mounted (section 5.4.6). Data output cannot therefore be written to a local drive. It can however be written by the script to the tmpfs, but data placed there is lost quite soon, namely during the `pivot_root` process that runs when the node-installer hands over control to the `init` process running from the local drive. However, if needed, the data can be placed on the local drive later by using the finalize script to copy it over from the tmpfs.

Due to this, and other reasons, a finalize script is easier to use for an administrator than an initialize script, and the use of the finalize script is therefore preferred.

### E.5.2 Ways Of Writing A Finalize Script To Configure The Destination Nodes

#### Basic Example—Copying A File To The Image

For a finalize script (section 5.4.11), which runs just before switching from using the ramdrive to using the local hard drive, the local hard drive is mounted under `/localdisk`. Data can therefore be written to the local hard drive if needed, but is only persistent until a reboot, when it gets rewritten. For example, predetermined configuration files can be written from the NFS drive for a particular node, or they can be written from an image prepared earlier and now running on the node at this stage, overwriting a node-installer configuration:

**Example**

```bash
#!/bin/bash
```

© Bright Computing, Inc.
Example initialize And finalize Scripts

This technique is used in a finalize script example in section 3.15.4, except that an append operation is used instead of a copy operation, to overcome a network issue by modifying a network configuration file slightly.

There are three important considerations for most finalize scripts:

1. Running A Finalize Script Without exit 0 Considered Harmful

Failed Finalize Script Logic Flow: For a default configuration without a finalize script, if PXE boot fails from the network during node provisioning, the node then goes on to attempt booting from the local drive via iPXE (section 5.1.2).

However, if the configuration has a finalize script, such as in the preceding example, and if the finalize script fails, then the failure is passed to the node installer.

Avoiding Remote Node Hang During A Finalize Script: exit 0 Recommended: If the node-installer fails, then no attempt is made to continue booting, and the node remains hung at that stage. This is usually undesirable, and can also make remote debugging of a finalize script annoying.

Adding an exit 0 to the end of the finalize script is therefore recommended, and means that an error in the script will still allow the node-installer to continue with an attempt to boot from the local drive.

Debugging Tips When A Node Hangs During A Finalize Script: If there is a need to understand the failure, then if the node-installer hangs, the administrator can ssh into the node into the node-installer environment, and run the finalize script manually to debug it. Once the bug has been understood, the script can be copied over to the appropriate location in the head node, for nodes or categories.

Additional aid in understanding a failure may be available by looking through the node-installer logs. The debug mode for the node-installer can be enabled by setting debug=true instead of debug=false in the file /cm/node-installer/scripts/node-installer.conf.

2. Protecting A Configuration File Change From Provisioning Erasure With excludelistupdate

In the preceding example, the finalize script saves a file /etc/myapp.conf to the destination nodes.

To protect such a configuration file from erasure, its file path must be covered in the second sublist in the excludelistupdate list (section 5.6.1).

3. Finalize scripts cannot modify /proc, /sys, and /dev filesystems of end result on node directly.

The /proc, /sys, and /dev filesystems are unmounted after the finalize script is run before pivoting into the root filesystem under the /localdisk directory, which means any changes made to them are simply discarded. To change values under these filesystems on the node, an rc.local file inside the software image can be used.

For example, if swappiness is to be set to 20 via the /proc filesystem, one way to do it is to set it in the rc.local file:

Example

```bash
# cat /cm/images/<image-name>/etc/rc.local | grep -v "^#" | grep .
echo 20 > /proc/sys/vm/swappiness
exit 0
# chmod 755 /cm/images/<image-name>/etc.d/rc.local # must be made executable
```
The preceding way of using `rc.local` set to run a command to modify the image just for illustration. A better way to get the same result in this case would be to not involve `rc.local`, but to add a line within the `/cm/images/<image-name>/etc/sysctl.conf` file:

```
vm.swappiness = 20
```

### Copying A File To The Image—Decision Based On Detection

Detection within a basic `finalize` script is useful extra technique. The `finalize` script example of section 3.15.4 does detection too, to decide if a configuration change is to be done on the node or not.

A further variation on a `finalize` script with detection is a script selecting from a choice of possible configurations. A symlink is set to one of the possible configurations based on hardware detection or detection of an environment variable. The environment variable can be a node parameter or similar, from the table in section E.3. If it is necessary to overwrite different nodes with different configurations, then the previous `finalize` script example might become something like:

#### Example

```bash
#!/bin/bash
if [[ $CMD_HOSTNAME = node00[1-7] ]]
  then ln -s /etc/myapp.conf.first /localdisk/etc/myapp.conf
fi
if [[ $CMD_HOSTNAME = node01[5-8] ]]
  then ln -s /etc/myapp.conf.second /localdisk/etc/myapp.conf
fi
if [[ $CMD_HOSTNAME = node02[3-6] ]]
  then ln -s /etc/myapp.conf.third /localdisk/etc/myapp.conf
fi
```

In the preceding example, the configuration file in the image has several versions: `/etc/myapp.conf.first|second|third`. Nodes node001 to node007 are configured with the first version, nodes node015 to node018 with the second version, and nodes node023 to node026 with the third version. It is convenient to add more versions to the structure of this decision mechanism.

### Copying A File To The Image—With Environmental Variables Evaluated In The File

Sometimes there can be a need to use the CMDaemon environment variables within a `finalize` script to specify a configuration change that depends on the environment.

For example a special service may need a configuration file, `test`, that requires the hostname `myhost`, as a parameter=value pair:

#### Example

```
SPECIALSERVICEPARAMETER=myhost
```

Ideally the placeholder value `myhost` would be the hostname of the node rather than the fixed value `myhost`. Conveniently, the CMDaemon environment variable `CMD_HOSTNAME` has the name of the host as its value.

So, inside the configuration file, after the administrator changes the host name from its placeholder name to the environment variable:

```
SPECIALSERVICE=${CMD_HOSTNAME}
```

then when the node-installer runs the `finalize` script, the file could be modified in-place by the `finalize` script, and `${CMD_HOSTNAME}` be substituted by the actual hostname.

A suitable `finalize` Bash script, which runs an in-line Perl substitution, is the following:

```
# Use the CMD_HOSTNAME environment variable.
SPECIALSERVICE=${CMD_HOSTNAME}
```

© Bright Computing, Inc.
Example initialize And finalize Scripts

#!/bin/bash
perl -p -i -e 's/${([^{}]+)}/defined $ENV{$1} ? $ENV{$1} : $&/eg' /localdisk/some/directory/file

Here, /some/directory/file means that, if for example the final configuration file path for the node is to be /var/spool/test then the file name should be set to /localdisk/var/spool/test inside the finalize script.

The finalize script replaces all lines within the file that have environment variable names of the form:

`PARAMETER=${<environment variable name>}`

with the value of that environment variable. Thus, if `<environment variable name>` is `CMD_HOSTNAME`, then that variable is replaced by the name of the host.

E.5.3 Restricting The Script To Nodes Or Node Categories

As mentioned in section 2.1.3, node settings can be adjusted within a category. So the configuration changes to `ifcfg-eth0` is best implemented per node by accessing and adjusting the finalize script per node if only a few nodes in the category are to be set up like this. If all the nodes in a category are to be set up like this, then the changes are best implemented in a finalize script accessed and adjusted at the category level. Accessing the scripts at the node and category levels is covered in section E.2.

People used to normal object inheritance behavior should be aware of the following when considering category level and node level finalize scripts:

With objects, a node item value overrules a category level value. On the other hand, finalize scripts, while treated in an analogous way to objects, cannot always inherit properties from each other in the precise way that true objects can. Thus, it is possible that a finalize script run at the node level may not have anything to do with what is changed by running it at the category level. However, to allow it to resemble the inheritance behavior of object properties a bit, the node-level finalize script, if it exists, is always run after the category-level script. This gives it the ability to “overrule” the category level.
F.1  Slurm

Slurm is a GPL-licensed workload management system and developed largely at Lawrence Livermore National Laboratory. The name was originally an acronym for Simple Linux Utility for Resource Management, but the acronym is deprecated because it no longer does justice to the advanced capabilities of Slurm.

The Slurm service and outputs are normally handled using the Bright View or cmsh front end tools for CMDaemon (section 7.4).

From the command line, direct Slurm commands that may sometimes come in useful include the following:

- `sacct`: used to report job or job step accounting information about active or completed jobs.

  **Example**

  ```
  # sacct -j 43 -o jobid,AllocCPUs,NCPUS,NNodes,NTasks,ReqCPUs
  JobID  AllocCPUs  NCPUS  NNodes  NTasks  ReqCPUs
  ------  ---------  -----  ------  -------  -------
  43     1         1      1       1
  ```

- `salloc`: used to allocate resources for a job in real time. Typically this is used to allocate resources and spawn a shell. The shell is then used to execute srun commands to launch parallel tasks.

- `sattach`: used to attach standard input, output, and error plus signal capabilities to a currently running job or job step. One can attach to and detach from jobs multiple times.

- `sbatch`: used to submit a job script for later execution. The script typically contains one or more srun commands to launch parallel tasks.

- `sbcast`: used to transfer a file from local disk to local disk on the nodes allocated to a job. This can be used to effectively use diskless compute nodes or provide improved performance relative to a shared filesystem.

- `scancel`: used to cancel a pending or running job or job step. It can also be used to send an arbitrary signal to all processes associated with a running job or job step.

- `scontrol`: the administrative tool used to view and/or modify Slurm state. Note that many scontrol commands can only be executed as user root.

  **Example**

© Bright Computing, Inc.


```bash
[fred@bright82 ~]$ scontrol show nodes
NodeName=bright82 Arch=x86_64 CoresPerSocket=1
    CPUAlloc=0 CPUErr=0 CPUTot=1 CPULoad=0.00 Features=(null)
...
```

- **sinfo**: reports the state of partitions and nodes managed by Slurm. It has a wide variety of filtering, sorting, and formatting options.

  **Example**

  ```bash
  bright82:~ # sinfo -o "%9P %.5a %.10l %.6D %.6t %C %N"
  PARTITION AVAIL TIMELIMIT NODES STATE CPUS(A/I/O/T) NODELIST
dfq* up infinite 1 alloc 1/0/0/1 bright82
  ...
  ```

- **smap**: reports state information for jobs, partitions, and nodes managed by Slurm, but graphically displays the information to reflect network topology.

- **squeue**: reports the state of jobs or job steps. It has a wide variety of filtering, sorting, and formatting options. By default, it reports the running jobs in priority order and then the pending jobs in priority order.

  **Example**

  ```bash
  bright82:~ # squeue -o "%.18i %.9P %.8j %.8u %.2t %.10M %.6D %C %R"
  JOBID PARTITION NAME USER ST TIME NODES CPUS NODELIST(REASON)
  43 defq bash fred R 16:22 1 1 bright82
  ...
  ```

- **srun**: used to submit a job for execution or initiate job steps in real time. srun has a wide variety of options to specify resource requirements, including: minimum and maximum node count, processor count, specific nodes to use or not use, and specific node characteristics (so much memory, disk space, certain required features, etc.). A job can contain multiple job steps executing sequentially or in parallel on independent or shared nodes within the job's node allocation.

- **smap**: reports state information for jobs, partitions, and nodes managed by Slurm, but graphically displays the information to reflect network topology.

- **strigger**: used to set, get or view event triggers. Event triggers include things such as nodes going down or jobs approaching their time limit.

- **sview**: a graphical user interface to get and update state information for jobs, partitions, and nodes managed by Slurm.

There are man pages for these commands. Full documentation on Slurm is available online at: [http://slurm.schedmd.com/documentation.html](http://slurm.schedmd.com/documentation.html).

## F.2 Sun Grid Engine

Sun Grid Engine (SGE) is a workload management system that was originally made available under an Open Source license by Sun Microsystems. It forked off into various versions in 2010 and its future is unclear at the time of writing, but it remains in widespread use. Bright Cluster Manager 8.2 uses version 6.2 update 5 patch 2 of Grid Scheduler, which is a bugfix-patched version of the last SGE release from Sun Microsystems, and is made available on sourceforge at [http://gridscheduler.sourceforge.net/](http://gridscheduler.sourceforge.net/).

SGE services should be handled using CMDaemon, as explained in section 7.4. However SGE can break in obtuse ways when implementing changes, so the following notes are sometimes useful in getting a system going again:

© Bright Computing, Inc.
• The sge_qmaster daemon on the head node can be started or stopped using /etc/init.d/sgemaster.sge1 start|stop, or alternatively via qconf -{s|k}m.

• The sge_execd execution daemon running on each compute node accepts, manages, and returns the results of the jobs on the compute nodes. The daemon can be started or stopped via /etc/init.d/sgeexecd start|stop, or alternatively deregistered from qmaster via qconf -{s|k}s.

• To see why a queue is in an error state, qstat -explain E shows the reason. Queues in an error state can be cleared with a qmod -c <queue name>.

SGE can be configured and managed generally with the command line utility qconf, which is what most administrators become familiar with. A GUI alternative, qmon, is also provided.

SGE commands are listed below. The details of these are in the man page of the command and the SGE documentation.

• qalter: modify existing batch jobs
• qacct: show usage information from accounting data
• qconf: configure SGE
• qdel: delete batch jobs
• qhold: place hold on batch jobs
• qhost: display compute node queues, states, jobs
• qlogin: start login-based interactive session with a node
• qmake: distributed, parallel make utility
• qmod: suspend/enable queues and jobs
• qmon: configure SGE with an X11 GUI interface
• qping: check sge_qmaster and sge_execd status
• qquota: list resource quotas
• qrresub: create new jobs by copying existing jobs
• qrdel: cancel advance reservations
• qrls: release batch jobs from a held state
• qrsh: start rsh-based interactive session with node
• qrstat: show status of advance reservations
• qrsub: submit advanced reservation
• qselect: select queues based on argument values
• qsh: start sh interactive session with a node
• qstat: show status of batch jobs and queues
• qsub: submit new jobs (related: qalter, qrresub)
• qtcsh: start csh-based interactive session with a node
F.3 Torque

The following commands can be used to manage Torque:

Torque resource manager commands:

- `qalter`: alter batch job
- `qdel`: delete batch job
- `qhold`: hold batch jobs
- `qrls`: release hold on batch jobs
- `qstat`: show status of batch jobs
- `qsub`: submit job
- `qmgr`: batch policies and configurations manager
- `qenable`: enable input to a destination
- `qdisable`: disable input to a destination
- `tracejob`: trace job actions and states

These direct commands are capable of modifying properties that CMDaemon does not manage, as well as properties that CMDaemon does manage. CMDaemon however overwrites the properties that are managed by CMDaemon with values recalled from the CMDaemon database, typically within a few minutes, if the values are changed directly via the direct commands.

Further information on these and other commands is available in the appropriate man pages and on-line documentation at http://www.adaptivecomputing.com/resources/docs/. The Torque administrator manual is also available from there.

F.4 PBS Pro

The following commands can be used in PBS Pro to view queues:

```
qstat                query queue status
qstat -a             alternate form
qstat -r             show only running jobs
qstat -q             show available queues
qstat -rn            only running jobs, w/ list of allocated nodes
qstat -i             only idle jobs
qstat -u username    show jobs for named user
```

Other useful commands are:

```
tracejob id          show what happened today to job id
tracejob -n d id     search last d days
qmgr                 administrator interface to batch system
qterm                terminates queues (but cm starts pbs_server again)
pbnodes -a           list available worker nodes in queue
```

The commands of PBS Pro are documented in the `PBS Professional 11.0 Reference Guide`. There is further extensive documentation for PBS Pro administrators in the `PBS Professional 11.0 Administrator's Guide`. Both of these guides are available at the PBS Works website at http://wwwpbsworks.com/SupportDocuments.aspx.
This appendix describes the metrics (section G.1), health checks (section G.2), enummetrics (section 12.2.2), and actions (section G.4), along with their parameters, in a newly-installed cluster. Metrics, health checks, enummetrics, and actions can each be standalone scripts, or they can be built-ins. Standalone scripts can be those supplied with the system, or they can be custom scripts built by the administrator. Scripts often require environment variables (as described in section 3.3.1 of the Developer Manual). On success scripts must exit with a status of 0, as is the normal practice.

G.1 Metrics And Their Parameters

A list of metric names can be viewed, for example, for the head node, using cmsh as follows (section 12.5.3):

```
[bright82 ~/]# cmsh -c "monitoring measurable; list metric"
```

Metrics are listed in this section as four types:

- regular metrics (section G.1.1)
- NFS metrics (section G.1.2)
- monitoring system metrics (section G.1.3)
- GPU metrics (section G.1.4)
- Job metrics (section G.1.5)
- Prometheus metrics (section G.1.6)

G.1.1 Regular Metrics

| Table G.1.1: List Of Metrics
| ...continues

© Bright Computing, Inc.
<table>
<thead>
<tr>
<th>Metric</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AlertLevel</td>
<td>Indicates the healthiness of a device based on severity of events (section 12.2.8). The lower it is, the better. There are 3 parameters it can take:</td>
</tr>
<tr>
<td></td>
<td>• count: the number of active triggers</td>
</tr>
<tr>
<td></td>
<td>• maximum: the maximum alert level of all active triggers</td>
</tr>
<tr>
<td></td>
<td>• sum: the average alert level of all active triggers</td>
</tr>
<tr>
<td>AvgJobDuration</td>
<td>Average Job Duration of current jobs</td>
</tr>
<tr>
<td>BlockedProcesses</td>
<td>Blocked processes waiting for I/O</td>
</tr>
<tr>
<td>BufferMemory</td>
<td>System memory used for buffering</td>
</tr>
<tr>
<td>BytesRecv*</td>
<td>Bytes/s received†</td>
</tr>
<tr>
<td>BytesSent†</td>
<td>Bytes/s sent‡</td>
</tr>
<tr>
<td>CPUGuest</td>
<td>CPU time spent in guest mode.</td>
</tr>
<tr>
<td>CPUIdle</td>
<td>CPU time spent in idle mode.</td>
</tr>
<tr>
<td>CPUIrq</td>
<td>CPU time spent in servicing IRQ.</td>
</tr>
<tr>
<td>CPUNice</td>
<td>CPU time spent in nice mode.</td>
</tr>
<tr>
<td>CPUSoftIrq</td>
<td>CPU time spent in servicing soft IRQ.</td>
</tr>
<tr>
<td>CPUSteal</td>
<td>CPU time spent in steal mode.</td>
</tr>
<tr>
<td>CPUSystem</td>
<td>CPU time spent in system mode.</td>
</tr>
<tr>
<td>CPUUser</td>
<td>CPU time spent in user mode.</td>
</tr>
<tr>
<td>CPUWait</td>
<td>CPU time spent in I/O wait mode.</td>
</tr>
<tr>
<td>CacheMemory</td>
<td>System memory used for caching.</td>
</tr>
<tr>
<td>CompletedJobs</td>
<td>Successfully completed jobs</td>
</tr>
<tr>
<td>Cores</td>
<td>Number of cores for a node</td>
</tr>
<tr>
<td>CoresTotal</td>
<td>Total number of known cores for all nodes</td>
</tr>
<tr>
<td>CoresUp</td>
<td>Number of cores for all nodes marked as UP</td>
</tr>
<tr>
<td>CtxtSwitches*</td>
<td>Context switches/s</td>
</tr>
<tr>
<td>Current_1**</td>
<td>First current seen by BMC sensor, in amps (file: sample_ipmi)</td>
</tr>
<tr>
<td>Current_2***</td>
<td>Second current seen by BMC sensor, in amps (file: sample_ipmi)</td>
</tr>
<tr>
<td>DevicesClosed</td>
<td>Number of devices not marked as UP or DOWN</td>
</tr>
<tr>
<td>DevicesDown</td>
<td>Number of devices marked as DOWN</td>
</tr>
</tbody>
</table>

...continues
<table>
<thead>
<tr>
<th>Metric</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DevicesTotal</td>
<td>Total number of devices</td>
</tr>
<tr>
<td>DevicesUp</td>
<td>Number of devices in status UP. A node (head, regular, virtual, cloud) or GPU Unit is not classed as a device. A device can be an item such as a switch, PDU, chassis, or rack, if the item is enabled and configured for management.</td>
</tr>
<tr>
<td>DropRecv⁴</td>
<td>Packets/s received and dropped‡</td>
</tr>
<tr>
<td>DropSent⁴</td>
<td>Packets/s sent and dropped‡</td>
</tr>
<tr>
<td>EccDBitGPU**</td>
<td>Total number of double bit ECC errors (file: sample_gpu)</td>
</tr>
<tr>
<td>EccSBitGPU**</td>
<td>Total number of single bit ECC errors (file: sample_gpu)</td>
</tr>
<tr>
<td>ErrorsRecv⁴</td>
<td>Packets/s received with error‡</td>
</tr>
<tr>
<td>ErrorsSent⁴</td>
<td>Packets/s sent with error‡</td>
</tr>
<tr>
<td>EstimatedDelay</td>
<td>Estimated delay time to execute jobs</td>
</tr>
<tr>
<td>FailedJobs</td>
<td>Failed completed jobs</td>
</tr>
<tr>
<td>Forks*</td>
<td>Forked processes/s</td>
</tr>
<tr>
<td>FrameErrors⁴</td>
<td>Packet framing errors/s</td>
</tr>
<tr>
<td>FreeSpace</td>
<td>Free space for non-root user. Takes mount point as a parameter</td>
</tr>
<tr>
<td>GPUUnitsClosed</td>
<td>Number of GPU units not marked as UP or DOWN</td>
</tr>
<tr>
<td>GPUUnitsDown</td>
<td>Number of GPU units marked as DOWN</td>
</tr>
<tr>
<td>GPUUnitsTotal</td>
<td>Total number of GPU Units</td>
</tr>
<tr>
<td>GPUUnitsUp</td>
<td>Number of GPU units marked as UP</td>
</tr>
<tr>
<td>HardwareCorruptedMemory</td>
<td>Hardware corrupted memory detected by ECC</td>
</tr>
<tr>
<td>IOInProgress</td>
<td>I/O operations in progress‡</td>
</tr>
<tr>
<td>IOTime</td>
<td>I/O operations time in milliseconds/s‡</td>
</tr>
<tr>
<td>Interrupts</td>
<td>System interrupts</td>
</tr>
<tr>
<td>IpForwDatagrams*</td>
<td>Input IP datagrams/s to be forwarded</td>
</tr>
<tr>
<td>IpFragCreates*</td>
<td>IP datagram fragments/s generated</td>
</tr>
<tr>
<td>IpFragFails*</td>
<td>IP datagrams/s which needed to be fragmented but could not</td>
</tr>
<tr>
<td>IpFragOKs*</td>
<td>IP datagrams/s successfully fragmented</td>
</tr>
<tr>
<td>IpInAddrErrors*</td>
<td>Input datagrams/s discarded because the IP address in their header was not a valid address</td>
</tr>
<tr>
<td>IpInDelivers*</td>
<td>Input IP datagrams/s successfully delivered</td>
</tr>
<tr>
<td>IpInDiscards*</td>
<td>Input IP datagrams/s discarded</td>
</tr>
<tr>
<td>IpInHdrErrors*</td>
<td>Input IP datagrams/s discarded due to errors in their IP headers</td>
</tr>
</tbody>
</table>

...continues
Table G.1.1: List Of Metrics...continued

<table>
<thead>
<tr>
<th>Metric</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>IpInReceives*</td>
<td>Input IP datagrams/s, including ones with errors, received from all interfaces</td>
</tr>
<tr>
<td>IpInUnknownProtos*</td>
<td>Input IP datagrams/s received but discarded due to an unknown or unsupported protocol</td>
</tr>
<tr>
<td>IpOutDiscards*</td>
<td>Output IP datagrams/s discarded</td>
</tr>
<tr>
<td>IpOutNoRoutes*</td>
<td>Output IP datagrams/s discarded because no route could be found</td>
</tr>
<tr>
<td>IpOutRequests*</td>
<td>Output IP datagrams/s supplied to IP in requests for transmission</td>
</tr>
<tr>
<td>IpReasmOKs*</td>
<td>IP datagrams/s successfully re-assembled</td>
</tr>
<tr>
<td>IpReasmReqs*</td>
<td>IP fragments/s received needing re-assembly</td>
</tr>
<tr>
<td>LoadFifteen</td>
<td>Load average on 15 minutes</td>
</tr>
<tr>
<td>LoadFive</td>
<td>Load average on 5 minutes</td>
</tr>
<tr>
<td>LoadOne</td>
<td>Load average on 1 minute</td>
</tr>
<tr>
<td>MajorPageFaults*</td>
<td>Page faults/s that require I/O</td>
</tr>
<tr>
<td>MemoryAvailable</td>
<td>Available system memory</td>
</tr>
<tr>
<td>MemoryFree</td>
<td>Free system memory</td>
</tr>
<tr>
<td>MemoryTotal</td>
<td>Total system memory</td>
</tr>
<tr>
<td>MemoryUsed</td>
<td>Used system memory</td>
</tr>
<tr>
<td>MemoryUsed:cmd</td>
<td>Used system memory for process cmd</td>
</tr>
<tr>
<td>MergedReads*</td>
<td>Merged reads/s†</td>
</tr>
<tr>
<td>MergedWrites*</td>
<td>Merged writes/s†</td>
</tr>
<tr>
<td>NodesClosed</td>
<td>Number of nodes not marked as UP or DOWN</td>
</tr>
<tr>
<td>NodesDown</td>
<td>Number of nodes marked as DOWN</td>
</tr>
<tr>
<td>NodesInQueue</td>
<td>Number of nodes in the queue</td>
</tr>
<tr>
<td>NodesTotal</td>
<td>Total number of nodes</td>
</tr>
<tr>
<td>NodesUp</td>
<td>Number of nodes in status UP</td>
</tr>
<tr>
<td>OccupationRate</td>
<td>Cluster occupation rate—a normalized cluster load percentage. 100% means all cores on all nodes are fully loaded. The calculation is done as follows: LoadOne on each node is mapped to a value, calibrated so that LoadOne=1 corresponds to 100% per node. The maximum allowed for a node in the mapping is 100%. The average of these mappings taken over all nodes is the OccupationRate.</td>
</tr>
</tbody>
</table>

...continues
### Table G.1.1: List Of Metrics...continued

<table>
<thead>
<tr>
<th>Metric</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>PacketsRecv</td>
<td>Packets/s received‡</td>
</tr>
<tr>
<td>PacketsSent</td>
<td>Packets/s sent‡</td>
</tr>
<tr>
<td>PageFaults</td>
<td>Page faults/s</td>
</tr>
<tr>
<td>PageIn</td>
<td>Number of bytes the system has paged in from disk</td>
</tr>
<tr>
<td>PageOut</td>
<td>Number of bytes the system has paged out to disk</td>
</tr>
<tr>
<td>PageSwapIn</td>
<td>Number of bytes the system has swapped in from disk</td>
</tr>
<tr>
<td>PageSwapOut</td>
<td>Number of bytes the system has swapped out to disk</td>
</tr>
<tr>
<td>PDUBankLoad</td>
<td>Total PDU bank load, in amps</td>
</tr>
<tr>
<td>PDULoad</td>
<td>Total PDU phase load, in amps</td>
</tr>
<tr>
<td>PDUUptime</td>
<td>PDU uptime per second. I.e. ideally=1, but in practice has jitter effects</td>
</tr>
<tr>
<td>ProcessCount</td>
<td>Total number of all processes running in the OS</td>
</tr>
<tr>
<td>QueuedJobs</td>
<td>Queued jobs</td>
</tr>
<tr>
<td>ReadTime</td>
<td>Read time in milliseconds/s‡</td>
</tr>
<tr>
<td>Reads</td>
<td>Reads/s completed successfully‡</td>
</tr>
<tr>
<td>RunningJobs</td>
<td>Running jobs</td>
</tr>
<tr>
<td>RunningProcesses</td>
<td>Running processes</td>
</tr>
<tr>
<td>SMARTHDATemp</td>
<td>Temperature of a Hard Disk Assembly‡</td>
</tr>
<tr>
<td>SMARTReallocSecCnt</td>
<td>SMART reallocated sectors count‡</td>
</tr>
<tr>
<td>SMARTSeekErrRate</td>
<td>SMART seek errors/s‡</td>
</tr>
<tr>
<td>SMARTSeekTimePerf</td>
<td>SMART average seek time‡</td>
</tr>
<tr>
<td>SMARTSoftReadErrRate</td>
<td>SMART software read errors/s‡</td>
</tr>
<tr>
<td>SectorsRead</td>
<td>Sectors/s read successfully/s‡</td>
</tr>
<tr>
<td>SectorsWritten</td>
<td>Sectors/s written successfully/s‡</td>
</tr>
<tr>
<td>SwapFree</td>
<td>Free swap memory</td>
</tr>
<tr>
<td>SwapTotal</td>
<td>Total swap memory</td>
</tr>
<tr>
<td>SwapUsed</td>
<td>Used swap memory</td>
</tr>
<tr>
<td>SystemTime:cmd</td>
<td>System time used by CMDaemon</td>
</tr>
</tbody>
</table>

© Bright Computing, Inc.
<table>
<thead>
<tr>
<th>Metric</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>TcpCurrEstab</td>
<td>TCP connections that are either ESTABLISHED or CLOSE-WAIT</td>
</tr>
<tr>
<td>TcpInErrs&lt;sup&gt;*&lt;/sup&gt;</td>
<td>Input IP segments/s received in error</td>
</tr>
<tr>
<td>TcpRetransSegs&lt;sup&gt;*&lt;/sup&gt;</td>
<td>Total number of IP segments/s re-transmitted</td>
</tr>
<tr>
<td>ThreadsUsed:cmd</td>
<td>Threads used by CMDaemon</td>
</tr>
<tr>
<td>TotalCPUIdle</td>
<td>Cluster-wide core usage in idle tasks (sum of all CPUIdle metric percentages)</td>
</tr>
<tr>
<td>TotalCPUSystem</td>
<td>Cluster-wide core usage in system mode (sum of all CPUSystem metric percentages)</td>
</tr>
<tr>
<td>TotalCPUUser</td>
<td>Cluster-wide core usage in user mode (sum of all CPUUser metric percentages)</td>
</tr>
<tr>
<td>TotalMemoryFree</td>
<td>Cluster-wide total of memory free</td>
</tr>
<tr>
<td>TotalMemoryUsed</td>
<td>Cluster-wide total of memory used</td>
</tr>
<tr>
<td>TotalSwapFree</td>
<td>Cluster-wide total swap free</td>
</tr>
<tr>
<td>TotalSwapUsed</td>
<td>Cluster-wide total swap used</td>
</tr>
<tr>
<td>TotalUser</td>
<td>Total number of known users</td>
</tr>
<tr>
<td>TotalUserLogin</td>
<td>Total number of logged in users</td>
</tr>
<tr>
<td>UdpInDatagrams&lt;sup&gt;*&lt;/sup&gt;</td>
<td>Input UDP datagrams/s delivered to UDP users</td>
</tr>
<tr>
<td>UdpInErrors&lt;sup&gt;*&lt;/sup&gt;</td>
<td>Input UDP datagrams/s received that could not be delivered/s for other reasons (no port excl.)</td>
</tr>
<tr>
<td>UdpNoPorts&lt;sup&gt;*&lt;/sup&gt;</td>
<td>Received UDP datagrams/s for which there was no application at the destination port</td>
</tr>
<tr>
<td>UniqueUserLogin</td>
<td>Number of unique users logged in</td>
</tr>
<tr>
<td>Uptime&lt;sup&gt;*&lt;/sup&gt;</td>
<td>System uptime per second. Ie, ideally=1, but in practice has jitter effects</td>
</tr>
<tr>
<td>UsedSpace</td>
<td>Total used space by a mount point. Takes mount point as a parameter.</td>
</tr>
<tr>
<td>UserTime:cmd</td>
<td>User time used by CMDaemon</td>
</tr>
<tr>
<td>VirtualMemoryUsed:cmd</td>
<td>Virtual memory used by CMDaemon</td>
</tr>
<tr>
<td>WriteTime&lt;sup&gt;*&lt;/sup&gt;</td>
<td>Write time in milliseconds/s†</td>
</tr>
<tr>
<td>Writes&lt;sup&gt;*&lt;/sup&gt;</td>
<td>Writes/s completed successfully†</td>
</tr>
</tbody>
</table>

* Cumulative metric. I.e. the metric is derived from cumulative raw measurements taken at two different times, according to:

\[
\text{metric}_{	ext{time2}} = \frac{\text{measurement}_{	ext{time2}} - \text{measurement}_{	ext{time1}}}{\text{time2} - \text{time1}}
\]

** Standalone scripts, not built-ins.

If sampling from a head node, the script is in directory: `/cm/local/apps/cmd/scripts/metrics/`
G.1.2 NFS Metrics
The NFS metrics correspond to `nfsstat` output, and are shown in table G.1.2.

<table>
<thead>
<tr>
<th>NFS Metric</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>nfs_client.packet.packets</code></td>
<td>NFS client packets statistics: packets</td>
</tr>
<tr>
<td><code>nfs_client.packet.tcp</code></td>
<td>NFS client package statistics: TCP/IP packets</td>
</tr>
<tr>
<td><code>nfs_client.packet.tcpconn</code></td>
<td>NFS client package statistics: TCP/IP connections</td>
</tr>
<tr>
<td><code>nfs_client.packet.udp</code></td>
<td>NFS client package statistics: UDP packets</td>
</tr>
<tr>
<td><code>nfs_client.rpc.authrefrsh</code></td>
<td>NFS Client RPC statistics: authenticated refreshes to RPC server</td>
</tr>
<tr>
<td><code>nfs_client.rpc.calls</code></td>
<td>NFS Client RPC statistics: calls</td>
</tr>
<tr>
<td><code>nfs_client.rpc.retrans</code></td>
<td>NFS Client RPC statistics: re-transmissions</td>
</tr>
<tr>
<td><code>nfs_server.file.anon</code></td>
<td>NFS Server file statistics: anonymous access</td>
</tr>
<tr>
<td><code>nfs_server.file.lookup</code></td>
<td>NFS Server file statistics: look-ups</td>
</tr>
<tr>
<td><code>nfs_server.file.ncachedir</code></td>
<td>NFS Server file statistics: ncachedir</td>
</tr>
<tr>
<td><code>nfs_server.file.stale</code></td>
<td>NFS Server file statistics: stale files</td>
</tr>
<tr>
<td><code>nfs_server.packet.packets</code></td>
<td>NFS Server packet statistics: packets</td>
</tr>
<tr>
<td><code>nfs_server.packet.tcp</code></td>
<td>NFS Server packet statistics: TCP/IP packets</td>
</tr>
<tr>
<td><code>nfs_server.packet.tcpconn</code></td>
<td>NFS Server packet statistics: TCP/IP connections</td>
</tr>
<tr>
<td><code>nfs_server.packet.udp</code></td>
<td>NFS Server packet statistics: UDP packets</td>
</tr>
<tr>
<td><code>nfs_server.reply.hits</code></td>
<td>NFS Server reply statistics: hits</td>
</tr>
<tr>
<td><code>nfs_server.reply.misses</code></td>
<td>NFS Server reply statistics: misses</td>
</tr>
<tr>
<td><code>nfs_server.reply.nocache</code></td>
<td>NFS Server reply statistics: no cache</td>
</tr>
<tr>
<td><code>nfs_server.rpc.badauth</code></td>
<td>NFS Server RPC statistics: bad authentication</td>
</tr>
<tr>
<td><code>nfs_server.rpc.badrcalls</code></td>
<td>NFS Server RPC statistics: bad RPC requests</td>
</tr>
<tr>
<td><code>nfs_server.rpc.badclnt</code></td>
<td>NFS Server RPC statistics: badclnt</td>
</tr>
<tr>
<td><code>nfs_server.rpc.calls</code></td>
<td>NFS Server RPC statistics: all calls to NFS and NLM</td>
</tr>
<tr>
<td><code>nfs_server.rpc.xdrcall</code></td>
<td>NFS Server RPC statistics: malformed XDR calls</td>
</tr>
<tr>
<td><code>nfs_v3_client.access</code></td>
<td>NFSv3 client statistics: access</td>
</tr>
<tr>
<td><code>nfs_v3_client.fsinfo</code></td>
<td>NFSv3 client statistics: static file system information</td>
</tr>
</tbody>
</table>

...continues...
### Table G.1.2: NFS Metrics

<table>
<thead>
<tr>
<th>NFS Metric</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>nfs_v3_client_fsstat</td>
<td>NFSv3 client statistics: dynamic file system status</td>
</tr>
<tr>
<td>nfs_v3_client_getattr</td>
<td>NFSv3 client statistics: file system attributes</td>
</tr>
<tr>
<td>nfs_v3_client_lookup</td>
<td>NFSv3 client statistics: lookup</td>
</tr>
<tr>
<td>nfs_v3_client_pathconf</td>
<td>NFSv3 client statistics: configuration path</td>
</tr>
<tr>
<td>nfs_v3_client_read</td>
<td>NFSv3 client statistics: reads</td>
</tr>
<tr>
<td>nfs_v3_client_total</td>
<td>NFSv3 client statistics: total</td>
</tr>
<tr>
<td>nfs_v3_server_access</td>
<td>NFSv3 server statistics: access</td>
</tr>
<tr>
<td>nfs_v3_server_create</td>
<td>NFSv3 server statistics: create</td>
</tr>
<tr>
<td>nfs_v3_server_fsinfo</td>
<td>NFSv3 server statistics: static file system information</td>
</tr>
<tr>
<td>nfs_v3_server_fsstat</td>
<td>NFSv3 server statistics: dynamic file system information</td>
</tr>
<tr>
<td>nfs_v3_server_getattr</td>
<td>NFSv3 server statistics: file system attributes gets</td>
</tr>
<tr>
<td>nfs_v3_server_lookup</td>
<td>NFSv3 server statistics: file name look-ups</td>
</tr>
<tr>
<td>nfs_v3_server_mkdir</td>
<td>NFSv3 server statistics: directory creation</td>
</tr>
<tr>
<td>nfs_v3_server_null</td>
<td>NFSv3 server statistics: null operations</td>
</tr>
<tr>
<td>nfs_v3_server_pathconf</td>
<td>NFSv3 server statistics: retrieve POSIX information</td>
</tr>
<tr>
<td>nfs_v3_server_read</td>
<td>NFSv3 server statistics: reads</td>
</tr>
<tr>
<td>nfs_v3_server.readdirplus</td>
<td>NFSv3 server statistics: READDIRPLUS procedures</td>
</tr>
<tr>
<td>nfs_v3_server_readlink</td>
<td>NFSv3 server statistics: Symbolic link reads</td>
</tr>
<tr>
<td>nfs_v3_server_setattr</td>
<td>NFSv3 server statistics: file system attribute sets</td>
</tr>
<tr>
<td>nfs_v3_server_total</td>
<td>NFSv3 server statistics: total</td>
</tr>
<tr>
<td>nfs_v3_server_write</td>
<td>NFSv3 server statistics: writes</td>
</tr>
</tbody>
</table>

### G.1.3 Monitoring System Metrics

The monitoring system metrics correspond to metrics for the monitoring data storage itself. These are shown in table G.1.3.

### Table G.1.3: Monitoring System Metrics

<table>
<thead>
<tr>
<th>Monitoring System Metric</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mon::CacheGather::handled</td>
<td>Cache gathers handled/s</td>
</tr>
<tr>
<td>Mon::CacheGather::miss</td>
<td>Cache gathers missed/s</td>
</tr>
<tr>
<td>Mon::DataProcessor::handled</td>
<td>Data processors handled/s</td>
</tr>
<tr>
<td>Mon::DataProcessor::miss</td>
<td>Data processors missed /s</td>
</tr>
<tr>
<td>Mon::DataTranslator::handled</td>
<td>Data translators handled/s</td>
</tr>
</tbody>
</table>

...continues

© Bright Computing, Inc.
<table>
<thead>
<tr>
<th>Monitoring System Metric</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mon::DataTranslator::miss</td>
<td>Data translators missed/s</td>
</tr>
<tr>
<td>Mon::EntityMeasurableCache::handled</td>
<td>Measurable cache handled/s</td>
</tr>
<tr>
<td>Mon::EntityMeasurableCache::miss</td>
<td>Measurable cache missed/s</td>
</tr>
<tr>
<td>Mon::MeasurableBroker::handled</td>
<td>Measurable broker handled/s</td>
</tr>
<tr>
<td>Mon::MeasurableBroker::miss</td>
<td>Measurable broker missed/s</td>
</tr>
<tr>
<td>Mon::Replicate::Collector::handled</td>
<td>Replication collection handled/s</td>
</tr>
<tr>
<td>Mon::Replicate::Collector::miss</td>
<td>Replication collection missed/s</td>
</tr>
<tr>
<td>Mon::Replicate::Combiner::handled</td>
<td>Replication combiner handled/s</td>
</tr>
<tr>
<td>Mon::Replicate::Combiner::miss</td>
<td>Replication combiner missed/s</td>
</tr>
<tr>
<td>Mon::RepositoryAllocator::handled</td>
<td>Repository allocator handled/s</td>
</tr>
<tr>
<td>Mon::RepositoryAllocator::miss</td>
<td>Repository allocator missed/s</td>
</tr>
<tr>
<td>Mon::RepositoryTrim::handled</td>
<td>Repository trim handled/s</td>
</tr>
<tr>
<td>Mon::RepositoryTrim::miss</td>
<td>Repository trim missed/s</td>
</tr>
<tr>
<td>Mon::Storage::Engine::elements</td>
<td>Storage engine data elements, in total</td>
</tr>
<tr>
<td>Mon::Storage::Engine::size</td>
<td>Storage engine size, in bytes</td>
</tr>
<tr>
<td>Mon::Storage::Engine::usage</td>
<td>Storage engine usage</td>
</tr>
<tr>
<td>Mon::Storage::Message::elements</td>
<td>Storage message data elements, in total</td>
</tr>
<tr>
<td>Mon::Storage::Message::size</td>
<td>Storage message size in bytes</td>
</tr>
<tr>
<td>Mon::Storage::Message::usage</td>
<td>Storage message usage</td>
</tr>
<tr>
<td>Mon::Storage::RepositoryId::elements</td>
<td>Storage repository ID data elements, in total</td>
</tr>
<tr>
<td>Mon::Storage::RepositoryId::size</td>
<td>Storage repository ID, size, in bytes</td>
</tr>
<tr>
<td>Mon::TaskInitializer::handled</td>
<td>Task initializer handled/s</td>
</tr>
<tr>
<td>Mon::TaskInitializer::miss</td>
<td>Task initializer missed/s</td>
</tr>
<tr>
<td>Mon::TaskSampler::handled</td>
<td>Task sampler handled/s</td>
</tr>
<tr>
<td>Mon::TaskSampler::miss</td>
<td>Task sampler missed/s</td>
</tr>
<tr>
<td>Mon::Trigger::Actuator::handled</td>
<td>Trigger actuators handled/s</td>
</tr>
<tr>
<td>Mon::Trigger::Actuator::miss</td>
<td>Trigger actuators missed/s</td>
</tr>
<tr>
<td>Mon::Trigger::Dispatcher::handled</td>
<td>Trigger dispatchers handled/s</td>
</tr>
<tr>
<td>Mon::Trigger::Dispatcher::miss</td>
<td>Trigger dispatchers missed/s</td>
</tr>
<tr>
<td>Prometheus::DataTranslator::handled</td>
<td>DataTranslator queries handled. The DataTranslator is a translation layer between PromQL format and Bright Cluster Manager format sampling.</td>
</tr>
</tbody>
</table>

...continues
<table>
<thead>
<tr>
<th>Monitoring System Metric</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Prometheus::DataTranslator::miss</td>
<td>DataTranslator misses</td>
</tr>
</tbody>
</table>

### G.1.4 GPU Metrics

The GPU metrics correspond to metrics provided by the GPU devices. The device parameter for the GPU metric specifies the device slot number that the GPU uses. The parameter takes the form `gpu0`, `gpu1`, and so on. It is appended to the metric with a colon character. For example, the `gpu_ecc_dbe_agg` metric, if used with `gpu1`, is specified as:

**Example**

```
  gpu_ecc_dbe_agg:gpu1
```

The data producer (section 12.2.10) for the GPU metrics is `GPUSampler`.

The available GPU metrics for the K40 and V100 GPUs are listed in table G.1.4. The available GPU health checks for the K40 and V100 GPUs are listed in table G.2.1.

<table>
<thead>
<tr>
<th>GPU Metric</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>gpu_ecc_dbe_agg</td>
<td>Total double bit aggregate ECC errors</td>
</tr>
<tr>
<td>gpu_ecc_dbe_vol</td>
<td>Total double bit volatile ECC errors</td>
</tr>
<tr>
<td>gpu_ecc_sbe_agg</td>
<td>Total single bit aggregate ECC errors</td>
</tr>
<tr>
<td>gpu_ecc_sbe_vol</td>
<td>Total single bit volatile ECC errors</td>
</tr>
<tr>
<td>gpu_enforced_power_limit</td>
<td>GPU enforced power limit</td>
</tr>
<tr>
<td>gpu_fb_total</td>
<td>GPU framebuffer size</td>
</tr>
<tr>
<td>gpu_fb_used</td>
<td>Amount of GPU memory used</td>
</tr>
<tr>
<td>gpu_memory_clock</td>
<td>GPU memory clock</td>
</tr>
<tr>
<td>gpu_power_management_limit</td>
<td>GPU power management limit</td>
</tr>
<tr>
<td>gpu_power_usage</td>
<td>GPU power usage</td>
</tr>
<tr>
<td>gpu_shutdown_temp</td>
<td>GPU shutdown temperature</td>
</tr>
<tr>
<td>gpu_slowdown_temp</td>
<td>GPU slowdown temperature</td>
</tr>
<tr>
<td>gpu_sm_clock</td>
<td>GPU shader multiprocessor clock</td>
</tr>
<tr>
<td>gpu_temperature</td>
<td>GPU temperature</td>
</tr>
<tr>
<td>gpu_utilization:average</td>
<td>Average GPU utilization percentage. When the</td>
</tr>
<tr>
<td></td>
<td>average parameter replaces the usual device</td>
</tr>
<tr>
<td></td>
<td>parameter, it samples the average percentage</td>
</tr>
<tr>
<td></td>
<td>over all the GPU devices that are in use</td>
</tr>
</tbody>
</table>


**G.1 Metrics And Their Parameters**

**Table G.1.4: GPU Metrics...continued**

<table>
<thead>
<tr>
<th>GPU Metric</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>gpu_utilization:gpu0</td>
<td>GPU utilization percentage for GPU using slot 0.</td>
</tr>
<tr>
<td></td>
<td>This follows the usual format</td>
</tr>
</tbody>
</table>

**G.1.5 Job Metrics**

Job metrics are introduced in section 12.8.1.

**Basic Job Metrics**

The following table lists some of the most useful job metrics that Bright Cluster Manager can monitor and visualize. In the table, the text `<device>` denotes a block device name, such as sda.

**Table G.1.5.1: Basic Job Metrics**

<table>
<thead>
<tr>
<th>Job Metric</th>
<th>Description</th>
<th>Cgroup Source File</th>
</tr>
</thead>
<tbody>
<tr>
<td>blkio.time:&lt;device&gt;</td>
<td>Time job had I/O access to device</td>
<td>blkio.time_recursive</td>
</tr>
<tr>
<td>blkio.sectors:&lt;device&gt;</td>
<td>Sectors transferred to or from specific devices by a cgroup</td>
<td>blkio.sectors_recursive</td>
</tr>
<tr>
<td>blkio.io_service_read:&lt;device&gt;</td>
<td>Bytes read</td>
<td>blkio.io_service_bytes_recursive</td>
</tr>
<tr>
<td>blkio.io_service_write:&lt;device&gt;</td>
<td>Bytes written</td>
<td>blkio.io_service_bytes_recursive</td>
</tr>
<tr>
<td>blkio.io_service_sync:&lt;device&gt;</td>
<td>Bytes transferred synchronously</td>
<td>blkio.io_service_bytes_recursive</td>
</tr>
<tr>
<td>blkio.io_service_async:&lt;device&gt;</td>
<td>Bytes transferred asynchronously</td>
<td>blkio.io_service_bytes_recursive</td>
</tr>
<tr>
<td>blkio.io_wait_time_read:&lt;device&gt;</td>
<td>Total time spent waiting for service in the scheduler queues for I/O read operations</td>
<td>blkio.io_wait_time_recursive</td>
</tr>
<tr>
<td>blkio.io_wait_time_write:&lt;device&gt;</td>
<td>Total time spent waiting for service in the scheduler queues for I/O write operations</td>
<td>blkio.io_wait_time_recursive</td>
</tr>
</tbody>
</table>

...continues
### Job Metric Description Cgroup Source File

<table>
<thead>
<tr>
<th>Job Metric</th>
<th>Description</th>
<th>Cgroup Source File</th>
</tr>
</thead>
<tbody>
<tr>
<td>blkio.io_wait_time_sync:&lt;device&gt;</td>
<td>Total time spent waiting for service in the scheduler queues for I/O synchronous operations</td>
<td>blkio.io_wait_time_recursive</td>
</tr>
<tr>
<td>blkio.io_wait_time_async:&lt;device&gt;</td>
<td>Total time spent waiting for service in the scheduler queues for I/O asynchronous operations</td>
<td>blkio.io_wait_time_recursive</td>
</tr>
<tr>
<td>cpuacct.usage</td>
<td>Total CPU time consumed by all job processes</td>
<td>cpuacct.usage</td>
</tr>
<tr>
<td>cpuacct.stat.user</td>
<td>User CPU time consumed by all job processes</td>
<td>cpuacct.stat</td>
</tr>
<tr>
<td>cpuacct.stat.system</td>
<td>System CPU time consumed by all job processes</td>
<td>cpuacct.stat</td>
</tr>
<tr>
<td>memory.usage</td>
<td>Total current memory usage</td>
<td>memory.usage_in_bytes</td>
</tr>
<tr>
<td>memory.memsuw.usage</td>
<td>Sum of current memory plus swap space usage</td>
<td>memory.memsuw.usage_in_bytes</td>
</tr>
<tr>
<td>memory.memsuw.max_usage</td>
<td>Maximum amount of memory and swap space used</td>
<td>memory.memsuw.max_usage_in_bytes</td>
</tr>
<tr>
<td>memory.failcnt</td>
<td>How often the memory limit has reached the value set in</td>
<td>memory.failcnt</td>
</tr>
<tr>
<td>memory.memsuw.failcnt</td>
<td>How often the memory plus swap space limit has reached the value set in</td>
<td>memory.memsuw.failcnt</td>
</tr>
<tr>
<td>memory.swap</td>
<td>Total swap usage</td>
<td>memory</td>
</tr>
</tbody>
</table>

...continues
...continued

<table>
<thead>
<tr>
<th>Job Metric</th>
<th>Description</th>
<th>Cgroup Source File</th>
</tr>
</thead>
<tbody>
<tr>
<td>memory.cache</td>
<td>Total page cache, including tmpfs (shmem)</td>
<td>memory</td>
</tr>
<tr>
<td>memory.mapped_file</td>
<td>Size of memory-mapped mapped files, including tmpfs (shmem)</td>
<td>memory</td>
</tr>
<tr>
<td>memory.unevictable</td>
<td>Memory that cannot be reclaimed</td>
<td>memory</td>
</tr>
</tbody>
</table>

The third column in the table shows the precise source file name that is used when the value is retrieved. These files are all virtual files, and are created as the cgroup controllers are mounted to the cgroup directory. In this case several controllers are mounted to the same directory, which means that all the virtual files will show up in that directory, and in its associated subdirectories—job cgroup directories—when the job runs.

**Advanced Job Metrics**

The metrics in the preceding table are enabled by default. There are also over 40 other advanced metrics that can be enabled via the `Enable Advanced Metrics` property of the `jobmetricsettings` object:

```
[bright82->monitoring->setup[JobSampler]->jobmetricsettings]% show
Parameter            Value
------------------------------------------------------------------------
Enable Advanced Metrics no
Exclude Devices       loop,sr
Include Devices       
Include Metrics       
Revision              
Sampling Type         Both
```

```
[bright82->monitoring->setup[JobSampler]->jobmetricsettings]% set enableadvancedmetrics yes
[bright82->monitoring->setup*[JobSampler*]->jobmetricsettings*]% commit
```

The advanced job metrics are:

**Table G.1.5.2: Advanced Job Metrics**

<table>
<thead>
<tr>
<th>Advanced Job Metric</th>
<th>Description</th>
<th>Cgroup Source File</th>
</tr>
</thead>
<tbody>
<tr>
<td>blkio.io_service_time_read:&lt;device&gt;</td>
<td>Total time between request dispatch and request completion according to CFQ scheduler for I/O read operations</td>
<td>blkio.io_service_time_recursive</td>
</tr>
</tbody>
</table>

...continues
### Advanced Job Metric Description Cgroup Source File

<table>
<thead>
<tr>
<th>Advanced Job Metric</th>
<th>Description</th>
<th>Cgroup Source File</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>blkio.io_service_time_write:&lt;device&gt;</code></td>
<td>Total time between request dispatch and request completion according to CFQ scheduler for I/O write operations</td>
<td><code>blkio.io_service_time_recursive</code></td>
</tr>
<tr>
<td><code>blkio.io_service_time_sync:&lt;device&gt;</code></td>
<td>Total time between request dispatch and request completion according to CFQ scheduler for I/O synchronous operations</td>
<td><code>blkio.io_service_time_recursive</code></td>
</tr>
<tr>
<td><code>blkio.io_service_time_async:&lt;device&gt;</code></td>
<td>Total time between request dispatch and request completion according to CFQ scheduler for I/O asynchronous operations</td>
<td><code>blkio.io_service_time_recursive</code></td>
</tr>
<tr>
<td><code>blkio.io_serviced_read:&lt;device&gt;</code></td>
<td>Read I/O operations</td>
<td><code>blkio.io_serviced_recursive</code></td>
</tr>
<tr>
<td><code>blkio.io_serviced_write:&lt;device&gt;</code></td>
<td>Write I/O operations</td>
<td><code>blkio.io_serviced_recursive</code></td>
</tr>
<tr>
<td><code>blkio.io_serviced_sync:&lt;device&gt;</code></td>
<td>Synchronous I/O operations</td>
<td><code>blkio.io_serviced_recursive</code></td>
</tr>
<tr>
<td><code>blkio.io_serviced_async:&lt;device&gt;</code></td>
<td>Asynchronous I/O operations</td>
<td><code>blkio.io_serviced_recursive</code></td>
</tr>
<tr>
<td><code>blkio.io_merged_read:&lt;device&gt;</code></td>
<td>Number of BIOS requests merged into requests for I/O read operations</td>
<td><code>blkio.io_merged_recursive</code></td>
</tr>
<tr>
<td><code>blkio.io_merged_write:&lt;device&gt;</code></td>
<td>Number of BIOS requests merged into requests for I/O write operations</td>
<td><code>blkio.io_merged_recursive</code></td>
</tr>
<tr>
<td><code>blkio.io_merged_sync:&lt;device&gt;</code></td>
<td>Number of BIOS requests merged into requests for I/O synchronous operations</td>
<td><code>blkio.io_merged_recursive</code></td>
</tr>
</tbody>
</table>

...continues
### Advanced Job Metric Description

<table>
<thead>
<tr>
<th>Advanced Job Metric</th>
<th>Description</th>
<th>Cgroup Source File</th>
</tr>
</thead>
<tbody>
<tr>
<td>blkio.io_merged_async:&lt;device&gt;</td>
<td>Number of BIOS requests merged into requests for I/O asynchronous operations</td>
<td>blkio.io_merged_recursive</td>
</tr>
<tr>
<td>blkio.io_queued_read:&lt;device&gt;</td>
<td>Number of requests queued for I/O read operations</td>
<td>blkio.io_queued_recursive</td>
</tr>
<tr>
<td>blkio.io_queued_write:&lt;device&gt;</td>
<td>Number of requests queued for I/O write operations</td>
<td>blkio.io_queued_recursive</td>
</tr>
<tr>
<td>blkio.io_queued_sync:&lt;device&gt;</td>
<td>Number of requests queued for I/O synchronous operations</td>
<td>blkio.io_queued_recursive</td>
</tr>
<tr>
<td>blkio.io_queued_async:&lt;device&gt;</td>
<td>Number of requests queued for I/O asynchronous operations</td>
<td>blkio.io_queued_recursive</td>
</tr>
<tr>
<td>memory.rss</td>
<td>Anonymous and swap cache, not including tmpfs (shmem)</td>
<td>memory</td>
</tr>
<tr>
<td>memory.pgpgin</td>
<td>Number of pages paged into memory</td>
<td>memory</td>
</tr>
<tr>
<td>memory.pgpgout</td>
<td>Number of pages paged out of memory</td>
<td>memory</td>
</tr>
<tr>
<td>memory.active_anon</td>
<td>Anonymous and swap cache on active least-recently-used (LRU) list, including tmpfs (shmem)</td>
<td>memory</td>
</tr>
<tr>
<td>memory.inactive_anon</td>
<td>Anonymous and swap cache on inactive LRU list, including tmpfs (shmem)</td>
<td>memory</td>
</tr>
<tr>
<td>memory.active_file</td>
<td>File-backed memory on active LRU list</td>
<td>memory</td>
</tr>
<tr>
<td>memory.inactive_file</td>
<td>File-backed memory on inactive LRU list</td>
<td>memory</td>
</tr>
</tbody>
</table>

...continues
G.1.6 Prometheus Metrics

Prometheus metrics are introduced in section 12.9.1.

The data producers for Prometheus metrics are JobSampler, JobMetadataSampler, and some others.

<table>
<thead>
<tr>
<th>Prometheus Metric</th>
<th>Description (for a job, unless asterisked)</th>
</tr>
</thead>
<tbody>
<tr>
<td>job_blkio_io_merged</td>
<td>Number of BIOS requests merged into requests for I/O operations by a cgroup</td>
</tr>
<tr>
<td>job_blkio_io_queued</td>
<td>Number of requests queued for I/O operations by a cgroup</td>
</tr>
<tr>
<td>job_blkio_io_service_bytes</td>
<td>Reports the number of bytes transferred to or from specific devices by a cgroup as seen by the CFQ scheduler</td>
</tr>
<tr>
<td>job_blkio_io_service_bytes_total</td>
<td>Reports the number of bytes transferred to or from specific devices by a cgroup as seen by the CFQ scheduler (for all jobs)</td>
</tr>
<tr>
<td>job_blkio_io_service_time_seconds</td>
<td>Reports the total time in seconds between request dispatch and request completion for I/O operations on specific devices by a cgroup as seen by the CFQ scheduler</td>
</tr>
<tr>
<td>job_blkio_io_serviced</td>
<td>Reports the number of I/O operations performed on specific devices by a cgroup as seen by the CFQ scheduler</td>
</tr>
<tr>
<td>job_blkio_io_wait_time_seconds</td>
<td>Reports the total time I/O operations on specific devices by a cgroup spent waiting for service in the scheduler queues</td>
</tr>
<tr>
<td>job_blkio_sectors</td>
<td>Reports the number of sectors transferred to or from specific devices by a cgroup</td>
</tr>
<tr>
<td>Prometheus Metric</td>
<td>Description (for a job, unless asterisked)</td>
</tr>
<tr>
<td>-------------------</td>
<td>------------------------------------------</td>
</tr>
<tr>
<td>job_blkio_time_seconds</td>
<td>Reports the time that a cgroup had I/O access to specific devices</td>
</tr>
<tr>
<td>job_cpuacct_stat_system</td>
<td>System CPU time consumed by processes</td>
</tr>
<tr>
<td>job_cpuacct_stat_user</td>
<td>User CPU time consumed by processes</td>
</tr>
<tr>
<td>job_cpuacct_usage_seconds</td>
<td>CPU usage time consumed</td>
</tr>
<tr>
<td>job_memory_active_anon_bytes</td>
<td>Anonymous and swap cache on active least-recently-used (LRU) list, including tmpfs (shmem), in bytes</td>
</tr>
<tr>
<td>job_memory_active_file_bytes</td>
<td>File-backed memory on active LRU list, in bytes</td>
</tr>
<tr>
<td>job_memory_cache_bytes</td>
<td>Page cache, including tmpfs (shmem), in bytes</td>
</tr>
<tr>
<td>job_memory_failcnt</td>
<td>Reports the number of times that the memory limit has reached the value set in memory.limit_in_bytes</td>
</tr>
<tr>
<td>job_memory_hierarchical_memory_limit_bytes</td>
<td>Memory limit for the hierarchy that contains the memory cgroup, in bytes</td>
</tr>
<tr>
<td>job_memory_hierarchical_memsw_limit_bytes</td>
<td>Memory plus swap limit for the hierarchy that contains the memory cgroup, in bytes</td>
</tr>
<tr>
<td>job_memory_inactive_anon_bytes</td>
<td>Anonymous and swap cache on inactive LRU list, including tmpfs (shmem), in bytes</td>
</tr>
<tr>
<td>job_memory_inactive_file_bytes</td>
<td>File-backed memory on inactive LRU list, in bytes</td>
</tr>
<tr>
<td>job_memory_mapped_file_bytes</td>
<td>Size of memory-mapped mapped files, including tmpfs (shmem), in bytes</td>
</tr>
<tr>
<td>job_memory_memsw_failcnt</td>
<td>Reports the number of times that the memory plus swap space limit has reached the value set in memory.memsw.limit_in_bytes</td>
</tr>
<tr>
<td>job_memory_memsw_max_usage_bytes</td>
<td>Reports the maximum amount of memory and swap space used by processes in the cgroup, in bytes</td>
</tr>
<tr>
<td>job_memory_memsw_usage_bytes</td>
<td>Reports the sum of current memory usage plus swap space used by processes in the cgroup, in bytes</td>
</tr>
<tr>
<td>job_memory_pgpgin_bytes</td>
<td>Number of pages paged into memory</td>
</tr>
<tr>
<td>job_memory_pgpgout_bytes</td>
<td>Number of pages paged out of memory</td>
</tr>
<tr>
<td>job_memory_rss_bytes</td>
<td>Anonymous and swap cache, not including tmpfs (shmem), in bytes</td>
</tr>
</tbody>
</table>

...continues
### Table G.1.6: Prometheus Metrics...continued

<table>
<thead>
<tr>
<th>Prometheus Metric</th>
<th>Description (for a job, unless asterisked)</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>job_memory_swap_bytes</code></td>
<td>Swap usage, in bytes</td>
</tr>
<tr>
<td><code>job_memory_unevictable_bytes</code></td>
<td>Memory that cannot be reclaimed, in bytes,</td>
</tr>
<tr>
<td><code>job_memory_usage_bytes</code></td>
<td>Reports the total current memory usage by processes in the cgroup, in bytes</td>
</tr>
<tr>
<td><code>job_memory_usage_bytes_total</code></td>
<td>Reports the total current memory usage by processes in the cgroup, in bytes (for all jobs)</td>
</tr>
<tr>
<td><code>job_metadata_is_running</code></td>
<td>Returns 1 if the job metadata sampler is running a job</td>
</tr>
<tr>
<td><code>job_metadata_is_waiting</code></td>
<td>Returns 1 if the job metadata sampler is waiting a job</td>
</tr>
<tr>
<td><code>job_metadata_num_cpus</code></td>
<td>Number of CPUs that the job runs on</td>
</tr>
<tr>
<td><code>job_metadata_num_nodes</code></td>
<td>Number of nodes that the job runs on</td>
</tr>
<tr>
<td><code>job_metadata_running_seconds</code></td>
<td>Time the job has run</td>
</tr>
<tr>
<td><code>job_metadata_waiting_seconds</code></td>
<td>Time the job has been waiting to run</td>
</tr>
<tr>
<td><code>users_job_effective_cpu_seconds:1w</code></td>
<td>CPU seconds used by users over the past week</td>
</tr>
<tr>
<td><code>users_job_running_count:1w</code></td>
<td>Number of jobs run by users over the past week</td>
</tr>
<tr>
<td><code>users_job_waiting_seconds:1w</code></td>
<td>Time users have been waiting for jobs to run over the past week</td>
</tr>
<tr>
<td><code>users_job_wall_clock_seconds:1w</code></td>
<td>Time the jobs runs for users according to wall time over the past week</td>
</tr>
<tr>
<td><code>users_job_wasted_cpu_seconds:1w</code></td>
<td>CPU time wasted during users jobs over the past week</td>
</tr>
</tbody>
</table>

### G.1.7 Parameters For Metrics

Metrics have the parameters indicated by the left column in the following example:

**Example**

```
[bright82->monitoring->measurable[CPUUser]]% show
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>------------------</td>
<td>--------------------------------------------</td>
</tr>
<tr>
<td>Class</td>
<td>CPU</td>
</tr>
<tr>
<td>Consolidator</td>
<td>default (ProcStat)</td>
</tr>
<tr>
<td>Cumulative</td>
<td>yes</td>
</tr>
<tr>
<td>Description</td>
<td>CPU time spent in user mode</td>
</tr>
<tr>
<td>Disabled</td>
<td>no (ProcStat)</td>
</tr>
<tr>
<td>Gap</td>
<td>0 (ProcStat)</td>
</tr>
<tr>
<td>Maximal age</td>
<td>0s (ProcStat)</td>
</tr>
<tr>
<td>Maximal samples</td>
<td>4,096 (ProcStat)</td>
</tr>
<tr>
<td>Maximum</td>
<td>0</td>
</tr>
<tr>
<td>Minimum</td>
<td>0</td>
</tr>
<tr>
<td>Name</td>
<td>CPUUser</td>
</tr>
<tr>
<td>Parameter</td>
<td></td>
</tr>
<tr>
<td>Producer</td>
<td>ProcStat</td>
</tr>
<tr>
<td>Revision</td>
<td></td>
</tr>
<tr>
<td>Type</td>
<td>Metric</td>
</tr>
<tr>
<td>Unit</td>
<td>Jiffies</td>
</tr>
</tbody>
</table>

If the value is inherited from the producer, then it is shown in parentheses next to the value. An inherited value can be overwritten by setting it directly for the parameter of a measurable.
The meanings of the parameters are:

**Class:** A choice assigned to a metric. It can be an internal type, or it can be a standalone class type. A slash (/) is used to separate class levels. A partial list of the class values is:

- **CPU:** CPU-related
- **Disk:** Disk-related
- **Internal:** An internal metric
- **Internal/Monitoring/Service:** An internal metric for monitoring a service
- **Internal/Monitoring/Storage:** An internal metric for monitoring storage
- **Job:** Job metric
- **Memory:** Memory-related
- **Network:** Network-related
- **OS:** Operating-system-related
- **Process:** Process-related
- **Prometheus:** Prometheus-related
- **Total:** Total cluster-wide-related
- **Workload:** Workload-related

**Consolidator:** This is described in detail in sections 12.4.3 and 12.5.2

**Cumulative:** If set to no, then the raw value is treated as not cumulative (for example, `CoresUp`), and the raw value is presented as the metric value.

If set to yes, then the metric is treated as being cumulative, which means that a rate (per second) value is presented.

More explicitly: When set to yes, it means that the raw sample used to calculate the metric is expected to be cumulative, like, for example, the bytes-received counter for an Ethernet interface. This in turn means that the metric is calculated from the raw value by taking the difference in raw sample measurement values, and dividing it by the time period over which the raw values are sampled. Thus, for example:

- The bytes-received raw measurements, which accumulate as the packets are received, and are in bytes, and have Cumulative set to yes, and then have a corresponding metric, `BytesRecv`, with a value in bytes/second.
- The system uptime raw measurements, which accumulate at the rate of 1 second per second, and are in seconds, have Cumulative set to yes, and have a corresponding metric, `Uptime`, with a value that uses no units. Ideally, the metric has a value of 1, but in practice the measured value varies a little due to jitter.

**Description:** Description of the raw measurement used by the metric. Empty by default.

**Disabled:** If set to no (default) then the metric runs.

**Gap:** The number of samples that are allowed to be missed before a value of NaN is set for the value of the metric.

**Maximal age:** the maximum age of RLE samples that are kept. If Maximal age is set to 0 then the sample age is not considered. Units can be w, d, h, m, s (weeks, days, hours, minutes, seconds), with s as the default.

© Bright Computing, Inc.
Maximal samples: the maximum number of RLE samples that are kept. If Maximal samples is set to 0 then the number of sample age is not considered.

Maximum: the default minimum value the y-axis maximum will take in graphs plotted in Bright View.¹

Minimum: the default maximum value the y-axis minimum will take in graphs plotted in Bright View.¹

Name: The name given to the metric.

Parameter: Parameter used for this metric. For example, eth0 with the metric BytesRecv

Producer: The data producer that produces the metric

Revision: User-definable revision number for the object

Type: This can be one of metric, healthcheck, or enummetric

Unit: A unit for the metric. For example: B/s (bytes/second) for BytesRecv metric, or unit-less for the Uptime metric. A percent is indicated with %

¹To clarify the concept, if maximum=3, minimum=0, then a data-point with a y-value of 2 is plotted on a graph with the y-axis spanning from 0 to 3. However, if the data-point has a y-value of 4 instead, then it means the default y-axis maximum of 3 is re-sized to 4, and the y-axis will now span from 0 to 4.

© Bright Computing, Inc.
G.2 Health Checks And Their Parameters

A list of health checks can be viewed, for example, for the head node, using cmsh as follows (section 12.5.3):

```
# cmsh -c "monitoring measurable; list healthcheck"
```

Health checks (section 12.2.4) are listed and described in this section.

### G.2.1 Health Checks

Table G.2.1: List Of Health Checks

<table>
<thead>
<tr>
<th>Name</th>
<th>Query (script response is PASS/FAIL)</th>
</tr>
</thead>
<tbody>
<tr>
<td>ManagedServicesOk*</td>
<td>Are CMDaemon-monitored services all OK? If the response is FAIL, then at least one of the services being monitored is failing. The latesthealtdata -v command (section 12.6.3) should show which one(s). After correcting the problem with the service, a reset of the service is normally carried out (section 3.11, page 110).</td>
</tr>
<tr>
<td>Mon::Storage</td>
<td>Is space available for the monitoring system metrics (section G.1.3)?</td>
</tr>
<tr>
<td>beegfs</td>
<td>Is BeeGFS properly connected as expected to all the nodes? Unreachable nodes and failed routes are listed in InfoMessages.</td>
</tr>
<tr>
<td>cmsh</td>
<td>Is cmsh available?</td>
</tr>
<tr>
<td>defaultgateway</td>
<td>Is there a default gateway available?</td>
</tr>
<tr>
<td>dellnss</td>
<td>If running, is the Dell NFS Storage Solution healthy?</td>
</tr>
<tr>
<td>diskspace</td>
<td>Is there less local disk space available to non-root users than any of the space parameters specified? The space parameters can be specified as MB, GB, TB, or as percentages with %. The default severity of notices from this check is 10, when one space parameter is used. For more than one space parameter, the severity decreases by 10 for each space parameter, sequentially, down to 10 for the last space parameter. By default a space parameter of 10% is assumed. Another, also optional, non-space parameter, the filesystem mount point parameter, can be specified after the last space parameter to track filesystem space, instead of disk space. A metric-based alternative to tracking filesystem space changes is to use the built-in metric freespace (page 699) instead.</td>
</tr>
</tbody>
</table>

...continued
### Table G.2.1: List Of Health Checks...continued

<table>
<thead>
<tr>
<th>Name</th>
<th>Query (response is PASS/FAIL)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Examples:

- **diskspace 10%**
  - less than 10% space = FAIL, severity 10

- **diskspace 10% 20% 30%**
  - less than 30% space = FAIL, with severity levels as indicated:

<table>
<thead>
<tr>
<th>space left</th>
<th>severity</th>
</tr>
</thead>
<tbody>
<tr>
<td>10%</td>
<td>30</td>
</tr>
<tr>
<td>20%</td>
<td>20</td>
</tr>
<tr>
<td>30%</td>
<td>10</td>
</tr>
</tbody>
</table>

- **diskspace 10GB 20GB**
  - less than 20GB space = FAIL, severity 10
  - less than 10GB space = FAIL, severity 20

- **diskspace 10% 20% /var**
  - For the filesystem /var:
  - less than 20% space = FAIL, severity 10
  - less than 10% space = FAIL, severity 20

...continued
<table>
<thead>
<tr>
<th>Name</th>
<th>Query (response is PASS/FAIL)</th>
</tr>
</thead>
<tbody>
<tr>
<td>dmesg</td>
<td>Is dmesg output OK?</td>
</tr>
<tr>
<td></td>
<td>Regexes to parse the output can be constructed in the configuration file at /cm/local/apps/cmd/scripts/healthchecks/configfiles/dmesg.py</td>
</tr>
<tr>
<td>docker</td>
<td>Is Docker running OK?</td>
</tr>
<tr>
<td></td>
<td>Checks for Docker server availability and corruption, dead containers, proper endpoints</td>
</tr>
<tr>
<td>dockerregistry</td>
<td>Is the Docker registry running OK? Checks registry endpoint and registry availability</td>
</tr>
<tr>
<td>exports</td>
<td>Are all filesystems as defined by the cluster management system exported?</td>
</tr>
<tr>
<td>etcd</td>
<td>Are the core etcd processes of Kubernetes running OK? Checks endpoints and interfaces</td>
</tr>
<tr>
<td>failedprejob</td>
<td>Are there failed prejob health checks (section 7.8.2)? Here: yes = FAIL. By default, the job ID is saved under /cm/shared/apps/&lt;scheduler&gt;/var/cm:/</td>
</tr>
<tr>
<td></td>
<td>• On FAIL, in failedprejobs.</td>
</tr>
<tr>
<td></td>
<td>• On PASS, in allprejobs</td>
</tr>
<tr>
<td>failover</td>
<td>Is the failover status OK?</td>
</tr>
<tr>
<td>gpu_health_driver</td>
<td>Is the Driver-related subsystem OK?</td>
</tr>
<tr>
<td>gpu_health_inforom</td>
<td>Is the Inforom OK?</td>
</tr>
<tr>
<td>gpu_health_mcu</td>
<td>Is the microcontroller unit OK?</td>
</tr>
<tr>
<td>gpu_health_mem</td>
<td>Is the memory subsystem OK?</td>
</tr>
<tr>
<td>gpu_health_nvlink</td>
<td>Is the NVLINK system OK?</td>
</tr>
<tr>
<td>gpu_health_PCIE</td>
<td>Is the PCIe system OK?</td>
</tr>
<tr>
<td>gpu_health_pmu</td>
<td>Is the power management unit OK?</td>
</tr>
<tr>
<td>gpu_health_power</td>
<td>Is the power OK?</td>
</tr>
<tr>
<td>gpu_health_sm</td>
<td>Is the streaming multiprocessor OK?</td>
</tr>
<tr>
<td>gpu_health_thermal</td>
<td>Is the temperature OK?</td>
</tr>
<tr>
<td>hpraid</td>
<td>Are the HP Smart Array controllers OK?</td>
</tr>
<tr>
<td>ib</td>
<td>Is the InfiniBand Host Channel Adapter working properly?</td>
</tr>
<tr>
<td></td>
<td>A configuration file for this health check is at /cm/local/apps/cmd/scripts/healthchecks/configfiles/ib.py</td>
</tr>
</tbody>
</table>

...continued
<table>
<thead>
<tr>
<th>Name</th>
<th>Query (response is PASS/FAIL)</th>
</tr>
</thead>
<tbody>
<tr>
<td>interfaces</td>
<td>Are the interfaces all up and OK?</td>
</tr>
<tr>
<td>ipmihealth</td>
<td>Is the BMC (IPMI or iLO) health OK? Uses the script sample_ipmi.</td>
</tr>
<tr>
<td>kuberneteschildnode</td>
<td>Are all Kubernetes child nodes up?</td>
</tr>
<tr>
<td>kubernetescomponent\</td>
<td>Are all expected agents and services up and running for active nodes?</td>
</tr>
<tr>
<td>tsstatus</td>
<td></td>
</tr>
<tr>
<td>kubernetesnodesstatus</td>
<td>Is the status for all Kubernetes nodes OK?</td>
</tr>
<tr>
<td>kubernetespodsstatus</td>
<td>Is the status for all pods OK?</td>
</tr>
<tr>
<td>ldap</td>
<td>Can the ID of the user be looked up with LDAP?</td>
</tr>
<tr>
<td>lustre</td>
<td>Is the Lustre filesystem running OK?</td>
</tr>
<tr>
<td>marathon</td>
<td>Is Marathon running OK?</td>
</tr>
<tr>
<td>messagequeue</td>
<td>Is the OpenStack Message Queue (RabbitMQ) running OK?</td>
</tr>
<tr>
<td>megaraid</td>
<td>Are the MegaRAID controllers OK?</td>
</tr>
<tr>
<td>mounts</td>
<td>Are all mounts defined in the fstab OK?</td>
</tr>
<tr>
<td>mysql</td>
<td>Is the status and configuration of MySQL correct?</td>
</tr>
<tr>
<td>node-hardware-profile</td>
<td>Is the specified node’s hardware configuration during health check use unchanged?</td>
</tr>
<tr>
<td>ntp</td>
<td>Is NTP synchronization happening?</td>
</tr>
<tr>
<td>oomkiller</td>
<td>Has the oomkiller process run?  Yes=FAIL. The oomkiller health check checks if the oomkiller process has run. The configuration file /cm/local/apps/cmd/scripts/healthchecks/configfiles/oomkiller.conf for the oomkiller health check can be configured to reset the response to PASS after one FAIL is logged, until the next oomkiller process runs. The processes killed by the oomkiller process are logged in /var/spool/cmd/save-oomkilleraction. A consideration of the causes and consequences of the killed processes is strongly recommended. A reset of the node is generally recommended.</td>
</tr>
</tbody>
</table>

...continued
### Table G.2.1: List Of Health Checks...continued

<table>
<thead>
<tr>
<th>Name</th>
<th>Query (response is PASS/FAIL)</th>
</tr>
</thead>
<tbody>
<tr>
<td>opalinkhealth</td>
<td>Are the quality and the integrity of the Intel OPA HFI link OK?</td>
</tr>
<tr>
<td>openstackaddremove</td>
<td>Is the addition and removal of objects in Keystone, Neutron, Nova, Cinder, Glance, and Heat</td>
</tr>
<tr>
<td>openstackagents</td>
<td>Are the agents (OpenStack service components) OK?</td>
</tr>
<tr>
<td>openstackauth</td>
<td>Is the authentication for Keystone, Neutron, Nova, Cinder, Glance, and Heat working OK?</td>
</tr>
<tr>
<td>openstackconfiguration</td>
<td>Are some basic configuration options for OpenStack working OK?</td>
</tr>
<tr>
<td>openstackdashboard</td>
<td>Is the Horizon dashboard login working OK?</td>
</tr>
<tr>
<td>openstackportstatus</td>
<td>Within OpenStack, are DHCP, router ports, and router gateway ports of the Neutron service all up?</td>
</tr>
<tr>
<td>openstackresponse</td>
<td>Is Keystone is responsive and are the endpoints of each service pingable?</td>
</tr>
<tr>
<td>openstackvmsok</td>
<td>Are the virtual machines under OpenStack running OK?</td>
</tr>
<tr>
<td>portchecker</td>
<td>Is the specified port on the specified host open for TCP (default) or UDP connections?</td>
</tr>
</tbody>
</table>
| rogueprocess          | Are the processes that are running legitimate (ie, not ‘rogue’)? Besides the FAIL/PASS/UNKNOWN response to CMDaemon, also returns a list of rogue process IDs to file descriptor 3 (InfoMessages), which the killprocess action (page 725) can then go ahead and kill. Illegitimate processes are processes that should not be running on the node. An illegitimate process is at least one of the following, by default:  
  • not part of the workload manager service or its jobs  
  • not a root- or system-owned process  
  • in the state Z, T, W, or X. States are described in the ps man pages in the section on “PROCESS STATE CODES” |

...continued
Table G.2.1: List Of Health Checks...continued

<table>
<thead>
<tr>
<th>Name</th>
<th>Query (response is PASS/FAIL)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rogue process criteria can be configured in the file /cm/local/apps/cmd/scripts/healthchecks/configfiles/rogueprocess.py within the software image. To implement a changed criteria configuration, the software image used by systems on which the health check is run should be updated (section 5.6). For example, using: cmsh -c &quot;device; imageupdate -c default -w&quot; for the default category of nodes.</td>
<td></td>
</tr>
<tr>
<td>schedulers</td>
<td>Are the queue instances of all schedulers on a node healthy?</td>
</tr>
<tr>
<td>smart</td>
<td>Is the SMART response healthy? The severities can be configured in the file /cm/local/apps/cmd/scripts/healthchecks/configfiles/smart.conf. By default, if a drive does not support the SMART commands and results in a &quot;Smart command failed&quot; info message for that drive, then the healthcheck is configured to give a PASS response. This is because the mere fact that the drive is a non-SMART drive should not be a reason to conclude that the drive is unhealthy. The info messages can be suppressed by setting a whitelist of the disks to be checked within /cm/local/apps/cmd/scripts/healthchecks/smart.</td>
</tr>
<tr>
<td>ssh2node</td>
<td>Is passwordless ssh root login, from head to a node that is up, working? Some details of its behavior are:</td>
</tr>
<tr>
<td></td>
<td>• If root ssh login to the head node has been disabled, then the health check fails on the head node.</td>
</tr>
<tr>
<td></td>
<td>• The healthcheck ssh2node only fails if ssh key-based access, to a node that is in the UP state, fails. The UP state is determined by whether CMDaemon is running on that node.</td>
</tr>
<tr>
<td></td>
<td>• If the regular node is in a DOWN state—which could be due to CMDaemon being down, or the node having been powered off gracefully, or the node suffering a sudden power failure—then the health check responds with a PASS. The idea here is to check key access, and decouple it from the node state.</td>
</tr>
</tbody>
</table>
### G.2 Health Checks And Their Parameters

<table>
<thead>
<tr>
<th>Name</th>
<th>Query (response is PASS/FAIL)</th>
</tr>
</thead>
<tbody>
<tr>
<td>swraid</td>
<td>Are the software RAID arrays healthy?</td>
</tr>
<tr>
<td>testhealthcheck</td>
<td>A health check script example for creating scripts, or setting a mix of PASS/FAIL/UNKNOWN responses. The source includes examples of environment variables that can be used, as well as configuration suggestions.</td>
</tr>
<tr>
<td>zookeeper</td>
<td>Are Zookeeper components healthy?</td>
</tr>
</tbody>
</table>

* built-ins, not standalone scripts.

If sampling from a head node, a standalone script is in directory:
/cm/local/apps/cmd/scripts/healthchecks/

If sampling from a regular node, a standalone script is in directory:
/cm/images/default-image/cm/local/apps/cmd/scripts/healthchecks/

### G.2.2 Parameters For Health Checks

Health checks have the parameters indicated by the left column in the example below:

**Example**

```
[myheadnode->monitoring->measurable]% show cmsh
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Class</td>
<td>Internal</td>
</tr>
<tr>
<td>Consolidator</td>
<td>- (cmsh)</td>
</tr>
<tr>
<td>Description</td>
<td>Checks whether cmsh is available, i.e. can we use cmsh for the default cluster?</td>
</tr>
<tr>
<td>Disabled</td>
<td>no (cmsh)</td>
</tr>
<tr>
<td>Gap</td>
<td>0 (cmsh)</td>
</tr>
<tr>
<td>Maximal age</td>
<td>0s (cmsh)</td>
</tr>
<tr>
<td>Maximal samples</td>
<td>4,096 (cmsh)</td>
</tr>
<tr>
<td>Name</td>
<td>cmsh</td>
</tr>
<tr>
<td>Parameter</td>
<td>HealthCheck</td>
</tr>
<tr>
<td>Producer</td>
<td>cmsh</td>
</tr>
<tr>
<td>Revision</td>
<td></td>
</tr>
</tbody>
</table>

If the value is inherited from the producer, then it is shown in parentheses next to the value. An inherited value can be overwritten by setting it directly for the parameter of a measurable.

The parameters are a subset of the parameters for metrics described in section G.1.7.
## G.3 Enummetrics

Table G.3: List Of Enummetrics

<table>
<thead>
<tr>
<th>Name</th>
<th>Query</th>
</tr>
</thead>
<tbody>
<tr>
<td>DeviceStatus</td>
<td>What is the status of the device? Possible values are:</td>
</tr>
<tr>
<td></td>
<td>• up</td>
</tr>
<tr>
<td></td>
<td>• down</td>
</tr>
<tr>
<td></td>
<td>• closed</td>
</tr>
<tr>
<td></td>
<td>• installing</td>
</tr>
<tr>
<td></td>
<td>• installer_failed</td>
</tr>
<tr>
<td></td>
<td>• installer_rebooting,</td>
</tr>
<tr>
<td></td>
<td>• installer_callinginit</td>
</tr>
<tr>
<td></td>
<td>• installer_unreachable</td>
</tr>
<tr>
<td></td>
<td>• installer_burning</td>
</tr>
<tr>
<td></td>
<td>• burning</td>
</tr>
<tr>
<td></td>
<td>• unknown</td>
</tr>
<tr>
<td></td>
<td>• opening</td>
</tr>
<tr>
<td></td>
<td>• going_down</td>
</tr>
<tr>
<td></td>
<td>• pending</td>
</tr>
<tr>
<td></td>
<td>• no data</td>
</tr>
</tbody>
</table>

© Bright Computing, Inc.
G.4 Actions And Their Parameters

G.4.1 Actions

Table G.4.1: List Of Actions

<table>
<thead>
<tr>
<th>Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Drain</td>
<td>Allows no new processes on a compute node from the workload manager.</td>
</tr>
<tr>
<td></td>
<td>This means that already running jobs are permitted to complete. Usage</td>
</tr>
<tr>
<td></td>
<td>Tip: Plan for undrain from another node becoming active</td>
</tr>
<tr>
<td>Send e-mail to</td>
<td>Sends mail using the mailserver that was set up during server configuration.</td>
</tr>
<tr>
<td>administrators</td>
<td>Default destination is root@localhost. The e-mail address that it is</td>
</tr>
<tr>
<td></td>
<td>otherwise sent to is specified by the recipient parameter for this action.</td>
</tr>
<tr>
<td>Event</td>
<td>Send an event to users with a connected client</td>
</tr>
<tr>
<td>ImageUpdate</td>
<td>Update the image on the node</td>
</tr>
<tr>
<td>PowerOff</td>
<td>Powers off, hard</td>
</tr>
<tr>
<td>PowerOn</td>
<td>Powers on, hard</td>
</tr>
<tr>
<td>PowerReset</td>
<td>Power reset, hard</td>
</tr>
<tr>
<td>Reboot</td>
<td>Reboot via the system, trying to shut everything down cleanly, and then</td>
</tr>
<tr>
<td></td>
<td>start up again</td>
</tr>
<tr>
<td>killprocess*</td>
<td>Kills processes listed in STDIN with KILL (-9) signal. Format: killprocess</td>
</tr>
<tr>
<td></td>
<td>&lt;PID1[,&lt;PID2&gt;,...]&gt;</td>
</tr>
<tr>
<td></td>
<td>This action is designed to work with rogueprocess (page 721)</td>
</tr>
<tr>
<td>remount*</td>
<td>remounts all defined mounts</td>
</tr>
<tr>
<td>testaction*</td>
<td>An action script example for users who would like to create their own</td>
</tr>
<tr>
<td></td>
<td>scripts. The source has helpful remarks about the environment variables</td>
</tr>
<tr>
<td></td>
<td>that can be used as well as tips on configuring it generally</td>
</tr>
<tr>
<td>Shutdown</td>
<td>Power off via system, trying to shut everything down cleanly</td>
</tr>
<tr>
<td>Undrain node</td>
<td>Allow processes to run on the node from the workload manager</td>
</tr>
</tbody>
</table>

* standalone scripts, not built-ins.

If running from a head node, the script is in directory: /cm/local/apps/cmd/scripts/actions/
If running from a regular node, the script is in directory: /cm/images/default-image/cm/local/apps/cmd/scripts/actions/

G.4.2 Parameters For A Monitoring Action

The default monitoring actions are listed in section 12.4.4.

All actions have in common the parameters shown by the left column, illustrated by the example below for the drain action:

Example

```
[myheadnode->monitoring->action]% show drain
Parameter                  Value
-------------------------------
Action                      Drain node from all WLM
Allowed time                no
Disable                     no
Name                        Drain
Revision                    
Run on                      Active
Type                        DrainAction
```

© Bright Computing, Inc.
Out of the full list of default actions, the actions with only the common parameter settings are:

- **Poweron**: Powers off the node
- **PowerOff**: Powers off the node
- **PowerReset**: Hard resets the node
- **Drain**: Drains the node (does not allow new jobs on that node)
- **Undrain**: Undrains the node (allows new jobs on that node)
- **Reboot**: Reboots node via the operating system.
- **Shutdown**: Shuts the node down via the operating system.
- **ImageUpdate**: Updates the node from the software image
- **Event**: Sends an event to users connected with cmsh or Bright View

### Extra Parameters For Some Actions

The following actions have extra parameters:

**Action of the type** `ScriptAction`:

- **killprocess**: A script that kills a specified process
- **testaction**: A test script
- **remount**: A script to remount all devices

The extra parameters for an action of type `ScriptAction` are:

- **Arguments**: List of arguments that are taken by the script
- **Node environment**: Does the script run in the node environment?
- **Script**: The script path
- **timeout**: Time within which the script must run before giving up

**Action of the type** `EmailAction`:

- **Send e-mail to administrators**: Sends an e-mail out, by default to the administrators

The extra parameters for an action of type `EmailAction` are:

- **All administrators**: sends the e-mail to the list of users in the Administrator e-mail setting in partition[base] mode
- **Info**: the body of the e-mail message
- **Recipients**: a list of recipients
Workload Manager
Configuration Files Updated By CMDaemon

This appendix lists workload manager configuration files changed by CMDaemon, events causing such change, and the file or property changed.

H.1 Slurm

<table>
<thead>
<tr>
<th>File/Property</th>
<th>Updates What?</th>
<th>Updated During</th>
</tr>
</thead>
<tbody>
<tr>
<td>/etc/slurm/slurm.conf</td>
<td>head node</td>
<td>Add/Remove/Update nodes, hostname change</td>
</tr>
<tr>
<td>/etc/slurm/slurmd.conf</td>
<td>head node</td>
<td>Add/Remove/Update nodes, hostname change</td>
</tr>
<tr>
<td>/etc/slurm/gres.conf</td>
<td>all nodes</td>
<td>Add/Remove/Update nodes</td>
</tr>
<tr>
<td>/etc/slurm/topology.conf</td>
<td>head node</td>
<td>Add/Remove/Update nodes</td>
</tr>
</tbody>
</table>

H.2 Grid Engine (SGE/UGE)

<table>
<thead>
<tr>
<th>File/Property</th>
<th>Updates What?</th>
<th>Updated During</th>
</tr>
</thead>
<tbody>
<tr>
<td>$ROOT/default/common/host_aliases</td>
<td>head node</td>
<td>hostname/domain change, failover</td>
</tr>
<tr>
<td>$ROOT/default/common/act_qmaster</td>
<td>head node</td>
<td>hostname/domain change, failover</td>
</tr>
</tbody>
</table>

H.3 Torque

<table>
<thead>
<tr>
<th>File/Property</th>
<th>Updates What?</th>
<th>Updated During</th>
</tr>
</thead>
<tbody>
<tr>
<td>$ROOT/spool/server_name</td>
<td>head node</td>
<td>hostname/domain change, failover</td>
</tr>
<tr>
<td>$ROOT/spool/torque.cfg</td>
<td>head node</td>
<td>hostname/domain change, failover</td>
</tr>
<tr>
<td>$ROOT/server_priv/acl_svr/acl_hosts</td>
<td>head node</td>
<td>hostname/domain change, failover</td>
</tr>
<tr>
<td>$ROOT/spool/server_priv/acl_svr/operators</td>
<td>head node</td>
<td>hostname change, failover</td>
</tr>
<tr>
<td>$ROOT/spool/server_priv/nodes</td>
<td>head node</td>
<td>Add/Remove/Update nodes</td>
</tr>
<tr>
<td>$ROOT/mom_priv/config</td>
<td>software image</td>
<td>hostname change, failover</td>
</tr>
</tbody>
</table>

H.4 PBS Pro

© Bright Computing, Inc.
<table>
<thead>
<tr>
<th>File/Property</th>
<th>Updates What?</th>
<th>Updated During</th>
</tr>
</thead>
<tbody>
<tr>
<td>/etc/pbs.conf</td>
<td>head node, software image</td>
<td>hostname/domain change, failover</td>
</tr>
<tr>
<td>$ROOT/server_priv/acl_svr/operators</td>
<td>head node</td>
<td>hostname change, failover</td>
</tr>
<tr>
<td>$ROOT/spool/server_priv/nodes</td>
<td>head node</td>
<td>Add/Remove/Update nodes</td>
</tr>
<tr>
<td>$ROOT/server_priv/acl_hosts</td>
<td>head node</td>
<td>Add/Remove/Update nodes</td>
</tr>
</tbody>
</table>

### H.5 LSF

<table>
<thead>
<tr>
<th>File/Property</th>
<th>Updates What?</th>
<th>Updated During</th>
</tr>
</thead>
<tbody>
<tr>
<td>$LSF_TOP/conf/lsf.conf</td>
<td>head node</td>
<td>hostname/domain change, failover</td>
</tr>
<tr>
<td>$LSF_TOP/conf/hosts</td>
<td>cloud-director</td>
<td>add/remove/update cloud nodes</td>
</tr>
<tr>
<td>$LSF_TOP/conf/lsbatch/&lt;clustename&gt;/</td>
<td>head node</td>
<td>add/remove/update nodes</td>
</tr>
<tr>
<td>configdir/lsb.queues</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$LSF_TOP/conf/lsbatch/&lt;clustename&gt;/</td>
<td>head node</td>
<td>add/remove/update nodes</td>
</tr>
<tr>
<td>configdir/lsb.hosts</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$LSF_TOP/conf/lsf.cluster.&lt;clustename&gt;</td>
<td>head node</td>
<td>add/remove/update nodes</td>
</tr>
</tbody>
</table>

The default value of $LSF_TOP in Bright Cluster Manager is /cm/shared/apps/lsf
Changing The LDAP Password

The administrator may wish to change the LDAP root password. This procedure has two steps:

- setting a new password for the LDAP server (section I.1), and
- setting the new password in cmd.conf (section I.2).

It is also a good idea to do some checking afterwards (section I.3).

I.1 Setting A New Password For The LDAP Server

An encrypted password string can be generated as follows:

```
[root@bright82 ~]# module load openldap
[root@bright82 ~]# slappasswd
New password: 
Re-enter new password: 
SSHAJ/3wyO+IqyAwhh8Q4obL8489CWJ1HpLg
```

The input is the plain text password, and the output is the encrypted password. The encrypted password is set as a value for the rootpw tag in the slapd.conf file on the head node:

```
[root@bright82 ~]# grep ^rootpw /cm/local/apps/openldap/etc/slapd.conf
rootpw:SSHAJ/3wyO+IqyAwhh8Q4obL8489CWJ1HpLg
```

The password can also be saved in plain text instead of as an SSHA hash generated with slappasswd, but this is considered insecure.

After setting the value for rootpw, the LDAP server is restarted:

```
[root@bright82 ~]# service ldap restart #Centos 6
[root@bright82 ~]# service slapd restart #Centos 7
```

I.2 Setting The New Password In cmd.conf

The new LDAP password (the plain text password that generated the encrypted password after entering the slappasswd command in section I.1) is set in cmd.conf. It is kept as clear text for the entry for the LDAPPass directive (Appendix C):

```
[root@bright82 ~]# grep LDAPPass /cm/local/apps/cmd/etc/cmd.conf
LDAPPass = "Mysecret1dappassw0rd"
```

CMDaemon is then restarted:

```
[root@bright82 ~]# service cmd restart
```
I.3 Checking LDAP Access

For a default configuration with user cmsupport and domain cm.cluster, the following checks can be run from the head node (some output truncated):

• anonymous access:

```
[root@bright82 ~]# ldapsearch -x
# extended LDIF
#
# LDAPv3
# base <dc=cm,dc=cluster> (default) with scope subtree
...
```

• root cn without a password (this should fail):

```
[root@bright82 ~]# ldapsearch -x -D 'cn=root,dc=cm,dc=cluster'
ldap_bind: Server is unwilling to perform (53)
   additional info: unauthenticated bind (DN with no password) disallowed
[root@bright82 ~]#
```

• root cn with a password (this should work):

```
[root@bright82 ~]# ldapsearch -x -D 'cn=root,dc=cm,dc=cluster' -w Mysecret1dappsw0rd
# extended LDIF
#
# LDAPv3
# base <dc=cm,dc=cluster> (default) with scope subtree
...
```
This appendix describes authorization tokens available for profiles. Profiles are introduced in Section 6.4:

<table>
<thead>
<tr>
<th>Service and token name</th>
<th>User can...</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Service: CMAuth</strong></td>
<td></td>
</tr>
<tr>
<td>GET_PROFILE_TOKEN</td>
<td>Retrieve list of profiles and profile properties</td>
</tr>
<tr>
<td>ADD_PROFILE_TOKEN</td>
<td>Add a new profile</td>
</tr>
<tr>
<td>MODIFY_PROFILE_TOKEN</td>
<td>Modify existing user profile</td>
</tr>
<tr>
<td>GET_CMSERVICES_TOKEN</td>
<td>Get a list of available CMDaemon services</td>
</tr>
<tr>
<td><strong>Service: CMCeph</strong></td>
<td></td>
</tr>
<tr>
<td>GET_CEPH_TOKEN</td>
<td>Get Ceph properties</td>
</tr>
<tr>
<td>ADD_CEPH_TOKEN</td>
<td>Add new Ceph configuration</td>
</tr>
<tr>
<td>UPDATE_CEPH_TOKEN</td>
<td>Update Ceph configuration</td>
</tr>
<tr>
<td>CEPH_KEYS_TOKEN</td>
<td>Manipulate Ceph keys</td>
</tr>
<tr>
<td>CEPH_CLUSTER_SERVICES_TOKEN</td>
<td>List Ceph services</td>
</tr>
<tr>
<td>CEPH_SET_DECOMISSION_TOKEN</td>
<td>Decommission Ceph nodes</td>
</tr>
<tr>
<td>CEPH_GET_DECOMISSION_TOKEN</td>
<td>Get decommissioning information</td>
</tr>
<tr>
<td>UPDATE_CEPH_STATE_TOKEN</td>
<td>Update Ceph state</td>
</tr>
<tr>
<td>GENERIC_CEPH_COMMAND_TOKEN</td>
<td>Run generic Ceph command</td>
</tr>
<tr>
<td><strong>Service: CMCert</strong></td>
<td></td>
</tr>
<tr>
<td>ISSUE_CERTIFICATE_TOKEN</td>
<td>Accept certificate request and issue signed certificate</td>
</tr>
<tr>
<td>GET_CERTIFICATE_REQUEST_TOKEN</td>
<td>List pending certificate requests</td>
</tr>
<tr>
<td>RECREATE_COMPONENT_CERTIFICATE_TOKEN</td>
<td>Recreate component certificate</td>
</tr>
</tbody>
</table>

...continues
<table>
<thead>
<tr>
<th>Service and token name</th>
<th>User can...</th>
</tr>
</thead>
<tbody>
<tr>
<td>REMOVE_CERTIFICATE_REQUEST_TOKEN</td>
<td>Cancel certificate request</td>
</tr>
<tr>
<td>GET_CERTIFICATE_TOKEN</td>
<td>Show certificate information</td>
</tr>
<tr>
<td>REVOKE_CERTIFICATE_TOKEN</td>
<td>Revoke a certificate</td>
</tr>
<tr>
<td>UNREVOKE_CERTIFICATE_TOKEN</td>
<td>Unrevoke a revoked certificate</td>
</tr>
<tr>
<td>GET_AUTOSIGN_TOKEN</td>
<td>Get information on certificate auto-signing CMDaemon’s private key</td>
</tr>
<tr>
<td>SET_AUTOSIGN_TOKEN</td>
<td>Enable certificate auto-signing with CMDaemon’s private key</td>
</tr>
<tr>
<td><strong>Service: CMCloud</strong></td>
<td></td>
</tr>
<tr>
<td>GET_CLOUD_PROVIDER_TOKEN</td>
<td>Get cloud provider information</td>
</tr>
<tr>
<td>ADD_CLOUD_PROVIDER_TOKEN</td>
<td>Add a new cloud provider</td>
</tr>
<tr>
<td>UPDATE_CLOUD_PROVIDER_TOKEN</td>
<td>Update cloud provider settings</td>
</tr>
<tr>
<td>EC2_ACCESS_STRING_TOKEN</td>
<td>Get/set Amazon EC2 access string</td>
</tr>
<tr>
<td>GET_CLOUD_REGION_TOKEN</td>
<td>Access Amazon EC2 region</td>
</tr>
<tr>
<td>GET_CLOUD_AMI_TOKEN</td>
<td>Access Amazon EC2 AMI</td>
</tr>
<tr>
<td>GET_CLOUD_TYPE_TOKEN</td>
<td>Access Amazon instance type</td>
</tr>
<tr>
<td>GET_KERNEL_INITRD_MD5SUM_TOKEN</td>
<td>Retrieve MD5 sum of initial ramdisk</td>
</tr>
<tr>
<td>PUT_USERDATA_TOKEN</td>
<td>Set AWS user data in AWS</td>
</tr>
<tr>
<td>TERMINATE_NODE_TOKEN</td>
<td>Terminate cloud nodes</td>
</tr>
<tr>
<td>GET_AWS_KEY_TOKEN</td>
<td>Retrieve AWS key</td>
</tr>
<tr>
<td>SET_CLOUDDIRECTOR_TOKEN</td>
<td>Modify the properties of the cloud director</td>
</tr>
<tr>
<td>CLOUD_DIRECTOR_NEW_IP_TOKEN</td>
<td>Set the new External IP of the cloud director</td>
</tr>
<tr>
<td>GET_CONSOLE_OUTPUT_TOKEN</td>
<td>Retrieve the console output of the cloud director for debugging purposes</td>
</tr>
<tr>
<td>SET_CLOUDERRORS_TOKEN</td>
<td>Get the static IP list of the cloud nodes</td>
</tr>
<tr>
<td>GET_CLOUD_VIRTUAL_NETWORK_INTERFACES_TOKEN</td>
<td>Get list of EC2 VPN interfaces (e.g. tun0)</td>
</tr>
<tr>
<td>SEND_CLOUD_STORAGE_ACTION_TOKEN</td>
<td>Send cloud storage action</td>
</tr>
<tr>
<td>VPC_MAGIC_CALL_TOKEN</td>
<td>(Internal use only)</td>
</tr>
<tr>
<td>ALLOCATE_STATIC_IP_TOKEN</td>
<td>Allocate static IP to cloud director</td>
</tr>
<tr>
<td>UNALLOCATE_STATIC_IP_TOKEN</td>
<td>De-allocate static IP</td>
</tr>
<tr>
<td>ALLOCATE_CLOUD_VIRTUAL_NETWORK_INTERFACE_TOKEN</td>
<td>Allocate virtual cloud network interface</td>
</tr>
<tr>
<td>UNALLOCATE_CLOUD_VIRTUAL_NETWORK_INTERFACE_TOKEN</td>
<td>De-allocate virtual cloud network interface</td>
</tr>
<tr>
<td>ATTACH_CLOUD_VIRTUAL_NETWORK_INTERFACE_TOKEN</td>
<td>Start a VPN tunnel to a cloud node</td>
</tr>
</tbody>
</table>

...continues
<table>
<thead>
<tr>
<th>Service and token name</th>
<th>User can...</th>
</tr>
</thead>
<tbody>
<tr>
<td>DETACH_CLOUD_VIRTUAL_NETWORK_INTERFACE</td>
<td>Stop a VPN tunnel to a cloud node</td>
</tr>
<tr>
<td>ADD_CLOUD_JOB_DESCRIPTION_TOKEN</td>
<td>Add cloud job description</td>
</tr>
<tr>
<td>GET_CLOUD_JOB_DESCRIPTION_TOKEN</td>
<td>Get cloud job description</td>
</tr>
<tr>
<td>GET_ALL_CLOUD_JOB_DESCRIPTION_TOKEN</td>
<td>Get all cloud job description</td>
</tr>
<tr>
<td>SUBMIT_CLOUD_JOB_DESCRIPTION_TOKEN</td>
<td>Submit cloud job description</td>
</tr>
<tr>
<td>UPDATE_CLOUD_JOB_DESCRIPTION_TOKEN</td>
<td>Update cloud job description</td>
</tr>
<tr>
<td><strong>Service: CMDevice</strong></td>
<td></td>
</tr>
<tr>
<td>SNMP_STRING_TOKEN</td>
<td>Get ethernet switch SNMP public string</td>
</tr>
<tr>
<td>CHASSIS_USER_PASSWORD_TOKEN</td>
<td>Get/set chassis username and password</td>
</tr>
<tr>
<td>BMC_USERNAME_PASSWORD_TOKEN</td>
<td>View/set BMC (e.g. HP ilo4, IPMI) username and password</td>
</tr>
<tr>
<td>GETDEVICE_TOKEN</td>
<td>View all device properties</td>
</tr>
<tr>
<td>GET_DEVICE_BY_PORT_TOKEN</td>
<td>View list of devices according to the ethernet switch port that they are connected to</td>
</tr>
<tr>
<td>ADD_DEVICE_TOKEN</td>
<td>Add a new device</td>
</tr>
<tr>
<td>UPDATE_DEVICE_TOKEN</td>
<td>Update device properties</td>
</tr>
<tr>
<td>GET_CATEGORY_TOKEN</td>
<td>Get list of categories</td>
</tr>
<tr>
<td>ADD_CATEGORY_TOKEN</td>
<td>Create new category</td>
</tr>
<tr>
<td>UPDATE_CATEGORY_TOKEN</td>
<td>Update a category property</td>
</tr>
<tr>
<td>GET_NODEGROUP_TOKEN</td>
<td>Get list of nodegroups</td>
</tr>
<tr>
<td>ADD_NODEGROUP_TOKEN</td>
<td>Add a new nodegroup</td>
</tr>
<tr>
<td>UPDATE_NODEGROUP_TOKEN</td>
<td>Update nodegroup properties (e.g. add a new member node)</td>
</tr>
<tr>
<td>GET_DEVICE_STATUS_TOKEN</td>
<td>Get device status (e.g. UP as well as status string e.g. restart required)</td>
</tr>
<tr>
<td>SET_DEVICE_STATUS_TOKEN</td>
<td>Set device status (only via RPC API calls)</td>
</tr>
<tr>
<td>POWER_ON_TOKEN</td>
<td>Power on a device using BMC or PDU power control</td>
</tr>
<tr>
<td>POWER_OFF_TOKEN</td>
<td>Power off a device</td>
</tr>
<tr>
<td>POWER_CYCLE_TOKEN</td>
<td>Power reset a device</td>
</tr>
<tr>
<td>POWER_STATUS_TOKEN</td>
<td>Get power status e.g on or off</td>
</tr>
<tr>
<td>POWER_RESULT_TOKEN</td>
<td>Get the result of the previous power command e.g. failed</td>
</tr>
<tr>
<td>SHUTDOWN_NODE_TOKEN</td>
<td>Shutdown a remote node managed by CMDaemon</td>
</tr>
<tr>
<td>REBOOT_NODE_TOKEN</td>
<td>Reboot a remote node</td>
</tr>
<tr>
<td>FORCE_RECONNECT_TOKEN</td>
<td>Force remote client to reconnect (RPC API)</td>
</tr>
<tr>
<td>NODE_IDENTIFY_TOKEN</td>
<td>Identify a node (RPC API, used by node installer)</td>
</tr>
</tbody>
</table>

...continues

© Bright Computing, Inc.
<table>
<thead>
<tr>
<th>Service and token name</th>
<th>User can...</th>
</tr>
</thead>
<tbody>
<tr>
<td>NODE_GET_MOUNTPOINTS_TOKEN</td>
<td>Get list of mountpoints defined for a node</td>
</tr>
<tr>
<td>PPING_TOKEN</td>
<td>Run parallel ping</td>
</tr>
<tr>
<td>BURN_STATUS_TOKEN</td>
<td>Get burn status</td>
</tr>
<tr>
<td>GET_BURN_LOG_TOKEN</td>
<td>Retrieve burn log</td>
</tr>
<tr>
<td>GET_SYNC_LOG_TOKEN</td>
<td>Get rsync provisioning log</td>
</tr>
<tr>
<td>GET_PORT_BY_MAC_TOKEN</td>
<td>Determine to which switch port a given MAC is connected to.</td>
</tr>
<tr>
<td>SYSINFO_COLLECTOR_TOKEN</td>
<td>Get information about a node (executes dmidecode)</td>
</tr>
<tr>
<td>UPDATE_CONFIG_TOKEN</td>
<td>Update node configuration</td>
</tr>
<tr>
<td>GET_LIVE_UCS_TOKEN</td>
<td>Get UCS live status</td>
</tr>
<tr>
<td>SET_LIVE_UCS_TOKEN</td>
<td>Set UCS live</td>
</tr>
<tr>
<td>VALIDATE_UCS_TOKEN</td>
<td>Validate UCS configuration</td>
</tr>
<tr>
<td>START_KVM_UCS_TOKEN</td>
<td>Start UCS KVM console</td>
</tr>
<tr>
<td>GET_UCS_LOG_TOKEN</td>
<td>Retrieve UCS log</td>
</tr>
<tr>
<td>CLEAR_UCS_LOG_TOKEN</td>
<td>Clear Cisco UCS log</td>
</tr>
<tr>
<td>CONVERT_XML_TO_UCS_TOKEN</td>
<td>Parse and serialize XML UCS configuration so that CMDaemon can use it</td>
</tr>
<tr>
<td>GET_EXCLUDE_LIST_TOKEN</td>
<td>Retrieve the various exclude lists</td>
</tr>
<tr>
<td>INSTALLER_REBOOT_REQUIRED_TOKEN</td>
<td>Set restart-required flag</td>
</tr>
<tr>
<td>(typically set by CMDaemon)</td>
<td></td>
</tr>
<tr>
<td>ADD_REMOTE_NODE_INSTALLER_INTERACTION_TOKEN</td>
<td>Add a node-installer interaction (Used by CMDaemon)</td>
</tr>
<tr>
<td>REMOVE_REMOTE_NODE_INSTALLER_INTERACTION_TOKEN</td>
<td>Remove a node installer interaction</td>
</tr>
<tr>
<td>GET_REMOTE_NODE_INSTALLER_INTERACTIONS_TOKEN</td>
<td>Get list of pending installer interactions (e.g. confirm full provisioning)</td>
</tr>
<tr>
<td>UPDATE_REMOTE_NODE_INSTALLER_INTERACTIONS_TOKEN</td>
<td>Update installer</td>
</tr>
<tr>
<td>GET_CLUSTER_STATUS_TOKEN</td>
<td>Get cluster status (RPC API, Internal)</td>
</tr>
<tr>
<td>SET_CLUSTER_STATUS_TOKEN</td>
<td>Set cluster status (RPC API, Internal)</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Service: CMGui</strong></td>
<td></td>
</tr>
<tr>
<td>GET_CLUSTER_OVERVIEW_TOKEN</td>
<td>Get cluster overview</td>
</tr>
<tr>
<td>GET_NODE_OVERVIEW_TOKEN</td>
<td>Get node overview</td>
</tr>
<tr>
<td>GET_NETSWITCH_OVERVIEW_TOKEN</td>
<td>Get switch overview</td>
</tr>
<tr>
<td>GET_PDU_OVERVIEW_TOKEN</td>
<td>Get PDU overview</td>
</tr>
<tr>
<td>GET_NODE_STATUS_TOKEN</td>
<td>Get node status</td>
</tr>
<tr>
<td>TOKEN</td>
<td></td>
</tr>
<tr>
<td>GET_CEPH_OVERVIEW_TOKEN</td>
<td>Get Ceph overview</td>
</tr>
<tr>
<td>GET_OPENSTACK_OVERVIEW_TOKEN</td>
<td>Get OpenStack overview</td>
</tr>
<tr>
<td>GET_OPENSTACK_TENANT_TOKEN</td>
<td>Get OpenStack VM</td>
</tr>
</tbody>
</table>

...continues
### Table J: List Of Tokens...continued

<table>
<thead>
<tr>
<th>Service and token name</th>
<th>User can...</th>
</tr>
</thead>
<tbody>
<tr>
<td>OVERVIEW_TOKEN</td>
<td>overview</td>
</tr>
</tbody>
</table>

**Service: CMJob**

- **GET_JOB_TOKEN**
  - Get list of jobs that are currently running
- **HOLD_JOB_TOKEN**
  - Place a job on hold
- **SUSPEND_JOB_TOKEN**
  - Suspend a job
- **RESUME_JOB_TOKEN**
  - Resume suspended job
- **RELEASE_JOB_TOKEN**
  - Release a held job
- **UPDATE_JOB_TOKEN**
  - Update job run-timer parameters
- **SUBMIT_JOB_TOKEN**
  - Submit a job using JSON
- **GET_JOBQUEUE_TOKEN**
  - Retrieve list of job queues and properties
- **UPDATE_JOBQUEUE_TOKEN**
  - Modify job queues
- **ADD_JOBQUEUE_TOKEN**
  - Add a new job queue
- **GET_PE_TOKEN**
  - Get list of SGE parallel environments
- **DRAIN_TOKEN**
  - Drain a node
- **DRAIN_OVERVIEW_TOKEN**
  - Obtain list of drained nodes

**Service: CMMain**

- **GET_LICENSE_INFO_TOKEN**
  - Retrieve information about BCM license
- **GET_VERSION_TOKEN**
  - Get CMDaemon version and revision
- **GET_SERVER_STATUS_TOKEN**
  - Headnode status (e.g. ACTIVE, BECOMEACTIVE etc.)
- **GET_CLUSTER_SETUP_TOKEN**
  - Get cluster configuration
- **PING_TOKEN**
  - TCP SYN ping managed devices
- **PCOPY_TOKEN**
  - Copy a specified file in parallel to a list of nodes
- **READ_FILE_TOKEN**
  - Read a text file. The text file will be serialized as a JSON object
- **SAVE_FILE_TOKEN**
  - Save a file on a remote node
- **UPDATE_SELF_TOKEN**
  - Update a category property
- **CMDAEMON_FAILOVER_TOKEN**
  - Set CMDaemon failover condition achieved
- **CMDAEMON_QUORUM_TOKEN**
  - Set CMDaemon quorum achieved
- **GENERIC_CALL_TOKEN**
  - Make a generic call
- **REPORT_CRITICAL_ERROR_TOKEN**
  - View critical error report
- **SET_SERVICESTATE_TOKEN**
  - Set the state of a service
- **GET_SERVICESTATE_TOKEN**
  - Get the state of a service
- **GET_BACKGROUND_TASKS_TOKEN**
  - Get background tasks
- **CANCEL_BACKGROUND_TASKS_TOKEN**
  - Cancel background tasks
- **CALL_EXTENSION_TOKEN**

**Service: CMMon**

...continues
<table>
<thead>
<tr>
<th>Service and token name</th>
<th>User can...</th>
</tr>
</thead>
<tbody>
<tr>
<td>GET_MONCONF_TOKEN</td>
<td>Get monitoring configuration settings</td>
</tr>
<tr>
<td>UPDATE_MONCONF_TOKEN</td>
<td>Update monitoring configuration settings</td>
</tr>
<tr>
<td>GET_METRIC_TOKEN</td>
<td>Get metric settings</td>
</tr>
<tr>
<td>UPDATE_METRIC_TOKEN</td>
<td>Update metric settings</td>
</tr>
<tr>
<td>ADD_METRIC_TOKEN</td>
<td>Add metric settings</td>
</tr>
<tr>
<td>GET_HEALTHCHECK_TOKEN</td>
<td>Get health check settings</td>
</tr>
<tr>
<td>UPDATE_HEALTHCHECK_TOKEN</td>
<td>Update health check settings</td>
</tr>
<tr>
<td>ADD_HEALTHCHECK_TOKEN</td>
<td>Add health check settings</td>
</tr>
<tr>
<td>GET_THRESHACTION_TOKEN</td>
<td>Get threshold action settings</td>
</tr>
<tr>
<td>UPDATE_THRESHACTION_TOKEN</td>
<td>Update threshold action settings</td>
</tr>
<tr>
<td>ADD_THRESHACTION_TOKEN</td>
<td>Add threshold action settings</td>
</tr>
<tr>
<td>GET_MONITORING_DATA_TOKEN</td>
<td>Get monitoring data settings</td>
</tr>
<tr>
<td>SIGNAL_THRESHOLD_EXCEEDED_TOKEN</td>
<td>See if signal threshold exceeded</td>
</tr>
<tr>
<td>MONITORING_INTERNAL_TOKEN</td>
<td>Set up internal monitoring</td>
</tr>
<tr>
<td>PREJOB_TOKEN</td>
<td>Set up prejob check</td>
</tr>
<tr>
<td>FAKE_MONITORING_TOKEN</td>
<td>Set up fake monitoring</td>
</tr>
<tr>
<td>ONDEMAND_TOKEN</td>
<td>Sample on demand</td>
</tr>
<tr>
<td>ONDEMAND_RESULT_TOKEN</td>
<td>Read sample results on demand.</td>
</tr>
<tr>
<td>GET_METRICCLASS_TOKEN</td>
<td>Get metric class settings</td>
</tr>
</tbody>
</table>

**Service: CMNet**

| GET_NETWORK_TOKEN              | Get network settings                              |
| ADD_NETWORK_TOKEN              | Add network settings                              |
| UPDATE_NETWORK_TOKEN           | Update network settings                           |

**CMOpenStack**

| GET_OPENSTACK_TOKEN            | Get OpenStack settings                            |
| ADD_OPENSTACK_TOKEN            | Add OpenStack settings                            |
| UPDATE_OPENSTACK_TOKEN         | Update OpenStack settings                         |
| OPENSTACK_USERNAMES_PASSWORDS_TOKEN | Get/set username and password settings           |
| OPENSTACK_MANAGE_USER_INSTANCES_TOKEN | Get/set user instances                           |
| OPENSTACK_VIEW_CONSOLE_LOG_TOKEN | View OpenStack console log                       |
| OPENSTACK_TERMINATE_USER_INSTANCES_TOKEN | Terminate OpenStack user instances               |
| OPENSTACK_EXECUTE_SETUP_TOKEN  | Execute OpenStack setup                           |
| OPENSTACK_REMOVE_SETUP_TOKEN   | Not execute OpenStack setup                       |

...continues
### Table J: List Of Tokens...continued

<table>
<thead>
<tr>
<th>Service and token name</th>
<th>User can...</th>
</tr>
</thead>
<tbody>
<tr>
<td>EXECUTION_TOKEN</td>
<td>User can...</td>
</tr>
<tr>
<td>GET_VNC_URL_TOKEN</td>
<td>Get OpenStack VNC URL</td>
</tr>
<tr>
<td><strong>Service: CMPart</strong></td>
<td></td>
</tr>
<tr>
<td>GET_PARTITION_TOKEN</td>
<td>Get partition settings</td>
</tr>
<tr>
<td>ADD_PARTITION_TOKEN</td>
<td>Add partition settings</td>
</tr>
<tr>
<td>UPDATE_PARTITION_TOKEN</td>
<td>Update partition settings</td>
</tr>
<tr>
<td>GET_RACK_TOKEN</td>
<td>Get rack settings</td>
</tr>
<tr>
<td>ADD_RACK_TOKEN</td>
<td>Add rack settings</td>
</tr>
<tr>
<td>UPDATE_RACK_TOKEN</td>
<td>Update rack settings</td>
</tr>
<tr>
<td>GET_SOFTWAREIMAGE_TOKEN</td>
<td>Get softwareimage settings</td>
</tr>
<tr>
<td>ADD_SOFTWAREIMAGE_TOKEN</td>
<td>Add softwareimage settings</td>
</tr>
<tr>
<td>UPDATE_SOFTWAREIMAGE_TOKEN</td>
<td>Update softwareimage settings</td>
</tr>
<tr>
<td>REMOVE_SOFTWAREIMAGE_TOKEN</td>
<td>Remove softwareimage settings</td>
</tr>
<tr>
<td>UPDATE_PROVISIONERS_TOKEN</td>
<td>Update provisioners settings</td>
</tr>
<tr>
<td>UPDATE_PROVISIONING_NODE_TOKEN</td>
<td>Update provisioning node</td>
</tr>
<tr>
<td>CMDEMON_FAILOVER_STATUS_TOKEN</td>
<td>Obtain status of failover</td>
</tr>
<tr>
<td><strong>Service: CMProc</strong></td>
<td></td>
</tr>
<tr>
<td>GET_PROCESS_TOKEN</td>
<td>Retrieve list of processes that are currently running on a device managed by CMDaemon</td>
</tr>
<tr>
<td>GET_SHARED_MEM_TOKEN</td>
<td>Get shared memory</td>
</tr>
<tr>
<td>GET_SEMAPHORE_TOKEN</td>
<td>Get semaphore</td>
</tr>
<tr>
<td>GET_MSGQUEUE_TOKEN</td>
<td>Get message queue status</td>
</tr>
<tr>
<td>CLEAN_IPC_TOKEN</td>
<td>Clear IPC state</td>
</tr>
<tr>
<td>SEND_SIGNAL_TOKEN</td>
<td>Send signal to a process</td>
</tr>
<tr>
<td>START_SHELL_TOKEN</td>
<td>Start SSH session</td>
</tr>
<tr>
<td>START_MINICOM_TOKEN</td>
<td>Start minicom serial session</td>
</tr>
<tr>
<td>EXEC_COMMAND_TOKEN</td>
<td>Execute a command on a headnode</td>
</tr>
<tr>
<td>NODE_EXEC_COMMAND_TOKEN</td>
<td>Remotely execute a command on a compute node</td>
</tr>
<tr>
<td>NODE_EXEC_COMMAND_MAINTENANCE_TOKEN</td>
<td>Execute a maintenance command (used by CMDaemon)</td>
</tr>
<tr>
<td>EXEC_INTERNAL_COMMAND_TOKEN</td>
<td>Execute internal command (defined in the source code, RPC API, internal)</td>
</tr>
<tr>
<td><strong>Service: CMProv</strong></td>
<td></td>
</tr>
<tr>
<td>GET_FSPART_TOKEN</td>
<td>Get FSPart (internal)</td>
</tr>
<tr>
<td>ADD_FSPART_TOKEN</td>
<td>Set FSPart (internal)</td>
</tr>
</tbody>
</table>

...continues
Table J: List Of Tokens...continued

<table>
<thead>
<tr>
<th>Service and token name</th>
<th>User can...</th>
</tr>
</thead>
<tbody>
<tr>
<td>UPDATE_FSPART_TOKEN</td>
<td>Update FSPart (internal)</td>
</tr>
<tr>
<td>REMOVE_FSPART_TOKEN</td>
<td>Remove FSPart (internal)</td>
</tr>
<tr>
<td>RUN_PROVISIONINGPROCESSORJOB_TOKEN</td>
<td>Start and run a provisioning job (nodes with a provisioning role)</td>
</tr>
<tr>
<td>UPDATE_PROVISIONINGPROCESSORJOB_TOKEN</td>
<td>Update status of running provisioning jobs (CMDaemon)</td>
</tr>
<tr>
<td>REQUEST_PROVISIONING_TOKEN</td>
<td>Request provisioning (nodes with a provisioning role)</td>
</tr>
<tr>
<td>MANAGE_RSYNC_DAEMON_TOKEN</td>
<td>Manage the rsync process (CMDaemon)</td>
</tr>
<tr>
<td>IMAGEUPDATE_TOKEN</td>
<td>Send image changes to nodes</td>
</tr>
<tr>
<td>UPDATEPROVISIONERS_TOKEN</td>
<td>Synchronize software images across provisioning systems</td>
</tr>
<tr>
<td>PROVISIONERS_STATUS_TOKEN</td>
<td>Check status of provisioners e.g. images are in sync</td>
</tr>
<tr>
<td>CANCEL_PROVISIONING_REQUEST_TOKEN</td>
<td>Cancel provisioning request</td>
</tr>
<tr>
<td>GRAB_IMAGEUPDATE_TOKEN</td>
<td>Grab changes from node to software image and vice versa</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Service: CMServ</strong></td>
<td>Get system service information</td>
</tr>
<tr>
<td>GET_OSSERVICE_TOKEN</td>
<td>Start system services (service foo start)</td>
</tr>
<tr>
<td>START_OSSERVICE_TOKEN</td>
<td>Stop system services</td>
</tr>
<tr>
<td>STOP_OSSERVICE_TOKEN</td>
<td>Call init (useful for the node-installer itself)</td>
</tr>
<tr>
<td>CALLINIT_OSSERVICE_TOKEN</td>
<td>Restart system services</td>
</tr>
<tr>
<td>RESTART_OSSERVICE_TOKEN</td>
<td>Reload system services</td>
</tr>
<tr>
<td>RELOAD_OSSERVICE_TOKEN</td>
<td>Reset system services</td>
</tr>
<tr>
<td>RESET_OSSERVICE_TOKEN</td>
<td></td>
</tr>
<tr>
<td><strong>Service: CMSession</strong></td>
<td>Retrieve session information</td>
</tr>
<tr>
<td>GET_SESSION_TOKEN</td>
<td>Register new nodes in a special CMDaemon session (node-installer)</td>
</tr>
<tr>
<td>REGISTER_NODE_SESSION_TOKEN</td>
<td>Terminate sessions</td>
</tr>
<tr>
<td>END_SESSION_TOKEN</td>
<td>Handle events</td>
</tr>
<tr>
<td>HANDLE_EVENT_TOKEN</td>
<td>Receive broadcast events</td>
</tr>
<tr>
<td>GET_BROADCAST_EVENTS_TOKEN</td>
<td></td>
</tr>
<tr>
<td><strong>Service: CMUser</strong></td>
<td>Retrieve user information</td>
</tr>
<tr>
<td>GET_USER_TOKEN</td>
<td>Add a new LDAP user</td>
</tr>
<tr>
<td>ADD_USER_TOKEN</td>
<td>Modify an existing LDAP user</td>
</tr>
<tr>
<td>UPDATE_USER_TOKEN</td>
<td>Retrieve group information</td>
</tr>
<tr>
<td>GET_GROUP_TOKEN</td>
<td>Add a new LDAP group</td>
</tr>
<tr>
<td>ADD_GROUP_TOKEN</td>
<td>Modify an existing LDAP group</td>
</tr>
<tr>
<td>UPDATE_GROUP_TOKEN</td>
<td></td>
</tr>
</tbody>
</table>
Understanding Consolidation

K.1 Introduction

Consolidation is discussed in the sections on using consolidation in the Monitoring chapter (sections 12.4.3 and 12.5.2).

However, it may be confusing to have the concept of consolidation discussed in the same place as the use of consolidation. Also, the algebra that appears in that discussion (page 483) may not appeal to people. There are many who would like an explanation that may be more intuitive, even if it is less rigorous.

Therefore, in this section a more informal and visual approach is taken to explain consolidation.

K.2 What Is Consolidation?

Consolidation is the compression of data, for data values that have been measured over a fixed interval. The compression is nothing particularly sophisticated. It is carried out by using some simple mathematical functions to the data points: the average, the maximum, or the minimum.

K.3 Raw Data And Consolidation

Suppose raw data is sampled every 2 minutes.

And the raw data values are consolidated every 10 minutes.

A visual representation of the data values available to the system is:

\[
\begin{array}{cccccccccc}
\text{--- time --->} \\
\text{raw:} & | & | & | & | & | & | & | & | \\
\text{consolidated:} & | & | & | & | & | & \\
\end{array}
\]

Here, every “|” indicates a data point, so that the visual shows 5 times as many raw data points as consolidated data values.

In the preceding visual it makes no sense to use consolidated data since the data values for raw data and consolidated data overlap. I.e., the more accurate raw data values exist for the entire period.

As time passes, the intention is to start dropping old raw data, to save space on the disk.

For example, for the first 20 minutes in the following visual, there are no longer raw data values available:

Example

\[
\begin{array}{cccccccccc}
\text{--- time --->} \\
\text{raw:} & | & | & | & | & | & | & | & | \\
\text{consolidated:} & | & | & | & | & | & \\
\end{array}
\]
But the consolidated data points for this period are still available to the system.
When the data values are plotted in Bright View graphs, periods without raw data values automatically have consolidated data values used.

So a combination of both data sources is used, which can be visually represented with:

Example

--- time --->
plot: | | | | | | | | | | | | | | |

That behavior holds true for cmsh too.
The behavior illustrated in the last visual assumes that the cluster has been UP for long enough that raw data is being dropped.

In this case, “long enough” means at least 7 days.

However, because RLE (Run Length Encoding) is used to compress the sampled monitoring data values on disk, this minimal “long enough” time can be (much) longer than 7 days. It depends on how much the measurable that is being sampled is changing as each sample is taken. if it is not changing, then RLE can compress over a longer time period.

For example, if a node has been up and reachable without issues for 1000 days, then the ssh2node health check raw data values would be PASS over that 1000 days. For the period from now to 7 days ago, the raw data values of PASS are kept as they are for now. However, for the period from 7 days ago to 1000 days ago, consolidation on the unchanging raw values means that only two values, namely the PASS value of 1000 days and the PASS value of 7 days ago, need to be retained, in order to have a totally accurate record of what the values were in that period.

On the other hand, the forks metric changes very quickly, and thus can do little RLE compression. That makes it a good choice for demonstrating the kind of output that the preceding visuals imply.

K.4 A Demonstration Of The Output

So, as a demonstration, the last 7 days for forks are now shown, with the data values in the middle elided:

Example

[bright82->device[bright82]]% dumpmonitoringdata -7d now forks

<table>
<thead>
<tr>
<th>Timestamp</th>
<th>Value</th>
<th>Info</th>
</tr>
</thead>
<tbody>
<tr>
<td>2018/10/17 10:30:00</td>
<td>2.76243 processes/s</td>
<td></td>
</tr>
<tr>
<td>2018/10/17 11:30:00</td>
<td>2.52528 processes/s</td>
<td></td>
</tr>
<tr>
<td>2018/10/17 12:30:00</td>
<td>2.53972 processes/s</td>
<td></td>
</tr>
<tr>
<td>...</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2018/10/24 10:42:00</td>
<td>2.66669 processes/s</td>
<td></td>
</tr>
<tr>
<td>2018/10/24 10:44:00</td>
<td>2.63333 processes/s</td>
<td></td>
</tr>
<tr>
<td>2018/10/24 10:46:00</td>
<td>2.64167 processes/s</td>
<td></td>
</tr>
</tbody>
</table>

The first part of the output shows samples listed every hour. These are the consolidated data values.
The last part of the output shows samples listed every 2 minutes. These are the raw data value values.

I.e.: consolidated data values are used beyond a certain time in the past.

If the administrator would like to explore this further, then displaying only consolidation values is possible in cmsh by using the --consolidationinterval option of the dumpmonitoringdata command:

Example
<table>
<thead>
<tr>
<th>Timestamp</th>
<th>Value</th>
<th>Info</th>
</tr>
</thead>
<tbody>
<tr>
<td>2019/01/07 11:39:06</td>
<td>2.65704 processes/s</td>
<td></td>
</tr>
<tr>
<td>2019/01/07 12:30:00</td>
<td>2.60111 processes/s</td>
<td></td>
</tr>
<tr>
<td>2019/01/07 13:30:00</td>
<td>2.58328 processes/s</td>
<td></td>
</tr>
<tr>
<td>...</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2019/01/07 18:09:06</td>
<td>2.59586 processes/s</td>
<td></td>
</tr>
<tr>
<td>2019/01/08 13:00:00</td>
<td>2.58953 processes/s</td>
<td></td>
</tr>
<tr>
<td>2019/01/09 06:06:06</td>
<td>2.58854 processes/s</td>
<td></td>
</tr>
</tbody>
</table>

© Bright Computing, Inc.
Node Execution Filters And Execution Multiplexers

Node execution filters and execution multiplexers define where data producers are executed on the nodes of a cluster, and what nodes are targeted to obtain the data.

This appendix explains how node execution filters and execution multiplexers work with the help of some explicit basic examples. The aim is to have the cluster administrator understand how they work and how to use them.

The reference cluster in this section is a 5-node cluster, made up of a head node (bright82) and 4 regular nodes (node001..node004). The commands run in this appendix are carried out during a cmsh session that continues on from the point that it left off earlier.

The terms “node execution filters” and “execution multiplexers” are commonly abbreviated to filters and multiplexers in this appendix.

A simple custom data producer script is created and used to explain some of the more-involved concepts of filters and multiplexers more clearly. The custom script is:

```
[root@bright82 ~]# cat /cm/shared/fm.sh
#!/bin/bash
echo $((RANDOM%100))
echo "Sampled on $(hostname) for $CMD_HOSTNAME" >&3
```

The preceding script outputs a random number, along with the host it is being run on ($\{hostname\}), and the host the metric is targeting ($\{CMD_HOSTNAME\}). The hosts that it is run on can be defined by filters, while the hosts that are targeted by the metric can be defined by multiplexers.

**The Term Multiplex:** The word “multiplex” can be confusing to system administrators. In electronics, the term multiplex implies that signals are being gathered from various inputs, and going into a main input.

Here the idea is applicable to the signals (samples) from the execution multiplexers (nodes where the samples are). The samples are multiplexed (gathered) from those nodes, to the node (or nodes) where the data producer is executing.

- The execution of the data producer is on the node (or nodes) defined by nodeexecutionfilter. The data producer execution nodes are the ones listed using the nodes command of cmsh.
- The nodes where the samples are obtained from are defined by the executionmultiplexer setting. Those multiplexer nodes can have their samples displayed as output using the samplenow command of cmsh.
L.1 Data Producers: Default Configuration For Running And Sampling

If there is no configuration defined for the data producer in the filters or multiplexers for that data producer, then each node runs a data producer on itself, and that data producer targets the node that it is running on.

For example, the existing dmesg data producer comprises the dmesg health check (section G.2.1) and by default has no filter or multiplexer defined for it. If an attempt is made to list any filter or multiplexer for dmesg, then by default there is no content under the table headings:

```
[root@bright82 ~]# cmsh
[bright82]% monitoring setup
[bright82->monitoring->setup]% nodeexecutionfilters dmesg; list; ..
```

```
<table>
<thead>
<tr>
<th>Type</th>
<th>Name (key)</th>
<th>Filter</th>
<th>Filter operation</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
```

```
[bright82->monitoring->setup]% executionmultiplexers dmesg; list; ..
```

```
<table>
<thead>
<tr>
<th>Name (key)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
</tr>
</tbody>
</table>
```

Another way of seeing that no such filters or multiplexers have been defined for dmesg could be by seeing that none are defined in its submodes:

```
[bright82->monitoring->setup]% show dmesg | grep submode
```

```
Execution multiplexer <0 in submode>
Node execution filters <0 in submode>
```

Most existing data producers have filters and multiplexers defined. The number of filters and multiplexers set per data producer can conveniently be viewed via list formatting:

Example

```
[bright82->monitoring->setup]% list -f name,nodeexecutionfilters,executionmultiplexer | more
```

```
name (key) nodeexecutionfilters executionmultiplexer
-------------------- -------------------- -------------------
AggregateNode <1 in submode> <1 in submode>
AggregatePDU <1 in submode> <1 in submode>
AlertLevel <1 in submode> <1 in submode>
CMDaemonState <0 in submode> <0 in submode>
Cassandra <1 in submode> <0 in submode>
ClusterTotal <1 in submode> <0 in submode>
DeviceState <1 in submode> <0 in submode>
... 
```

L.1.1 Nodes That Data Producers Are Running On By Default—The nodes Command

The nodes command shows which nodes the data producer runs on. By default, the data producer runs on all nodes, when nothing has been set explicitly, because each node runs the data producer for itself:

```
[bright82->monitoring->setup]% nodes dmesg
node001..node004,bright82
```

L.1.2 Nodes That Data Producers Target By Default—The samplenow Command

Nodes where samples are being obtained at can be seen using the samplenow command for the specified nodes.

Again, by default, each node is a target, because the target is same node that the dmesg data producer runs on:
In the outputs to `samplenow` displayed in this appendix, some columns are omitted for the sake of clarity.

The `-t node` option to `samplenow` expands to `-n node001..node004,bright82` for this reference cluster.

## L.2 Data Producers: Configuration For Running And Targeting

Filters and multiplexers define which nodes run the data producers, and which nodes are the targets for measurables.

The `fm.sh` script introduced on page 743 can be used to define several custom metrics according to what nodes run the script and what nodes are targeted by the script.

### L.2.1 Custom Metrics From The `fm.sh` Custom Script

Custom data producers of type `metric` are created in this section. These data producers comprise custom metrics, which are now set up with varying filtering and multiplexing definitions, to illustrate how the definitions work.

**The Metric `all_for_self`**

The metric from the data producer `all_for_self` can be set up with no filtering or multiplexing defined, as follows:

```
[bright82->monitoring->setup]% add metric all_for_self
[bright82->monitoring->setup*][all_for_self*]% set consolidator none
[bright82->monitoring->setup*][all_for_self*]% set script /cm/shared/fm.sh
[bright82->monitoring->setup*][all_for_self*]% set class Test
[bright82->monitoring->setup*][all_for_self*]% commit
```

The value for `class` is mandatory but arbitrary. It is an arbitrary grouping mechanism, which can be useful in Bright View for grouping folders in trees.

**Sampling results for the metric `all_for_self`:** With no filtering or multiplexing, the metric just runs everywhere by default, with the target of the running metric being itself too. The `Info` field output from `samplenow` shows this behavior in the script output:

```
[bright82->monitoring->setup][all_for_self]% exit
[bright82->monitoring->setup]% device samplenow -t node all_for_self
```

<table>
<thead>
<tr>
<th>Entity</th>
<th>Measurable</th>
<th>Type</th>
<th>Value</th>
<th>Age</th>
<th>Info</th>
</tr>
</thead>
<tbody>
<tr>
<td>node001</td>
<td>all_for_self</td>
<td>Test</td>
<td>2</td>
<td>0.08s</td>
<td>Sampled on node001 for node001</td>
</tr>
<tr>
<td>node002</td>
<td>all_for_self</td>
<td>Test</td>
<td>13</td>
<td>0.084s</td>
<td>Sampled on node002 for node002</td>
</tr>
<tr>
<td>node003</td>
<td>all_for_self</td>
<td>Test</td>
<td>97</td>
<td>0.08s</td>
<td>Sampled on node003 for node003</td>
</tr>
<tr>
<td>node004</td>
<td>all_for_self</td>
<td>Test</td>
<td>18</td>
<td>0.084s</td>
<td>Sampled on node004 for node004</td>
</tr>
<tr>
<td>bright82</td>
<td>all_for_self</td>
<td>Test</td>
<td>63</td>
<td>0.144s</td>
<td>Sampled on bright82 for bright82</td>
</tr>
</tbody>
</table>
The Metric `some_for_self`

The metric `some_for_self` can be set up with filtering set up for some nodes, and no multiplexing set up, as follows:

```
# on node001, node002 for itself
[bright82->monitoring->setup]% add metric some_for_self
[bright82->monitoring->setup*{some_for_self*}]% set consolidator none
[bright82->monitoring->setup*{some_for_self*}]% set class Test
[bright82->monitoring->setup*{some_for_self*}]% set script /cm/shared/fm.sh
[bright82->...*{some_for_self*}]% nodeexecutionfilters
[bright82->...*{some_for_self*}]->nodeexecutionfilters% add node some_nodes
[bright82->...*nodeexecutionfilters*{some_nodes*}]% set nodes node001 node002
[bright82->...*nodeexecutionfilters*{some_nodes*}]% show
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Filter operation</td>
<td>Include</td>
</tr>
<tr>
<td>Name</td>
<td>some_nodes</td>
</tr>
<tr>
<td>Nodes</td>
<td>node001, node002</td>
</tr>
<tr>
<td>Revision</td>
<td></td>
</tr>
<tr>
<td>Type</td>
<td>Node</td>
</tr>
</tbody>
</table>

```
[bright82->...*nodeexecutionfilters*{some_nodes*}]% commit
```

Sampling results for the metric `some_for_self`:

Only some filtering defined, and no multiplexing defined at all, means that the metric just runs on the filtered nodes, and targets only the nodes defined in filter too:

```
[bright82->...*nodeexecutionfilters*{some_nodes*}]% ..; ..; ..
[bright82->monitoring->setup]% device samplenow -t node some_for_self
```

<table>
<thead>
<tr>
<th>Entity</th>
<th>Measurable</th>
<th>Type</th>
<th>Value</th>
<th>Age</th>
<th>Info</th>
</tr>
</thead>
<tbody>
<tr>
<td>node001</td>
<td>some_for_self</td>
<td>Test</td>
<td>88</td>
<td>0.094s</td>
<td>Sampled on node001 for node001</td>
</tr>
<tr>
<td>node002</td>
<td>some_for_self</td>
<td>Test</td>
<td>98</td>
<td>0.075s</td>
<td>Sampled on node002 for node002</td>
</tr>
</tbody>
</table>

The Metric `from_head_for_some_others`

The metric `from_head_for_some_others` can be set up with filtering defined for the head node, and multiplexing defined for some other regular nodes (other than node001 and node002 here), as follows:

```
# on active head for node003, node004
[bright82->monitoring->setup]% add metric from_head_for_some_others
[bright82->monitoring->setup*{from_head_for_some_others*}]% set consolidator none
[bright82->monitoring->setup*{from_head_for_some_others*}]% set script /cm/shared/fm.sh
[bright82->monitoring->setup*{from_head_for_some_others*}]% set class Test
[bright82->...*{from_head_for_some_others*}]% nodeexecutionfilters
[bright82->...*{from_head_for_some_others*}]->nodeexecutionfilters% active
```

Added active resource filter
```
[bright82->...*{from_head_for_some_others*}]->nodeexecutionfilters% show active head node
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Filter operation</td>
<td>Include</td>
</tr>
<tr>
<td>Name</td>
<td>Active head node</td>
</tr>
<tr>
<td>Operator</td>
<td>OR</td>
</tr>
<tr>
<td>Resources</td>
<td>Active</td>
</tr>
<tr>
<td>Revision</td>
<td></td>
</tr>
<tr>
<td>Type</td>
<td>Resource</td>
</tr>
</tbody>
</table>

```
[bright82->monitoring->setup*{from_head_for_some_others*}]->nodeexecutionfilters% ..
```

© Bright Computing, Inc.
L.3 Replacing A Resource With An Explicit Node Specification

Within the filter Active head node, associated with the metric from_head_for_some_others, the resource object Active can be replaced with a node object instead, if the node is defined as being bright82. Doing this on a high-availability cluster where there is one active and one passive head node, would be unwise. However, doing this on the reference cluster for teaching purposes is of course absolutely fine because it helps make things a bit more concrete for the reader. The replacement can be carried out for the session as follows:

```
[bright82->monitoring->setup] % nodeexecutionfilters from_head_for_some_others
[bright82->...[from_head_for_some_others]->nodeexecutionfilters] % list
```

<table>
<thead>
<tr>
<th>Type</th>
<th>Name (key)</th>
<th>Filter</th>
<th>Filter operation</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

© Bright Computing, Inc.
### Node Execution Filters And Execution Multiplexers

<table>
<thead>
<tr>
<th>Resource</th>
<th>Active head node</th>
<th>Active</th>
<th>Include</th>
</tr>
</thead>
<tbody>
<tr>
<td>[bright82-&gt;...[from_head_for_some_others]-&gt;nodeexecutionfilters]</td>
<td>remove active head node</td>
<td></td>
<td></td>
</tr>
<tr>
<td>[bright82-&gt;...<em>[from_head_for_some_others</em>]-&gt;nodeexecutionfilters*]</td>
<td>add node head</td>
<td></td>
<td></td>
</tr>
<tr>
<td>[bright82-&gt;...<em>[from_head_for_some_others</em>]-&gt;nodeexecutionfilters*[head*]]</td>
<td>set nodes bright82</td>
<td></td>
<td></td>
</tr>
<tr>
<td>[bright82-&gt;...<em>[from_head_for_some_others</em>]-&gt;nodeexecutionfilters*[head*]]</td>
<td>show</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Parameter**

<table>
<thead>
<tr>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Include</td>
</tr>
<tr>
<td>head</td>
</tr>
<tr>
<td>bright82</td>
</tr>
<tr>
<td>Node</td>
</tr>
</tbody>
</table>

The sample results are the same kind of output, and the filter node used is the same. The target sample outputs are the same kind of output as before the replacement. The filter node on which the metric runs is also seen to be the same:

[bright82->monitoring->setup]% device samplenow -t node from_head_for_some_others

<table>
<thead>
<tr>
<th>Entity</th>
<th>Measurable</th>
<th>Type</th>
<th>Value</th>
<th>Age</th>
<th>Info</th>
</tr>
</thead>
<tbody>
<tr>
<td>node003</td>
<td>from_head_for_some_others</td>
<td>Test</td>
<td>11</td>
<td>0.081s</td>
<td>Sampled on bright82 for node003</td>
</tr>
<tr>
<td>node004</td>
<td>from_head_for_some_others</td>
<td>Test</td>
<td>12</td>
<td>0.07s</td>
<td>Sampled on bright82 for node004</td>
</tr>
</tbody>
</table>

[bright82->monitoring->setup]% nodes bright82

### L.4 Excessive Sampling

If another node is appended to the node in the filter, then a warning comes up.

[bright82->monitoring->setup]% nodeexecutionfilters from_head_for_some_others

Field Message

<table>
<thead>
<tr>
<th>Field</th>
<th>Message</th>
</tr>
</thead>
<tbody>
<tr>
<td>executionMultiplexers</td>
<td>Warning: Execution filters/multiplexers are set to run on multiple nodes for the same target. This likely means they are badly configured</td>
</tr>
</tbody>
</table>

The warning is there because the node execution filters are doing the same thing from different nodes for an execution multiplexer target, and duplicating monitoring execution is typically a waste of resources, and thus typically a mistake.

© Bright Computing, Inc.
However, the warnings are merely warnings, and not errors. So Bright Cluster Manager just goes ahead with setting up the filter/multiplex system according to what the administrator has specified. The nodes and sampling now commands now show:

```
[bright82->...[from_head_for_some_others]->nodeexecutionfilters]% ..;..
[bright82->monitoring->setup]% device samplenow -t node from_head_for_some_others
```

<table>
<thead>
<tr>
<th>Entity</th>
<th>Measurable</th>
<th>Type</th>
<th>Value</th>
<th>Age</th>
<th>Info</th>
</tr>
</thead>
<tbody>
<tr>
<td>node003</td>
<td>from_head_for_some_others</td>
<td>Test</td>
<td>3</td>
<td>0.151s</td>
<td>Sampled on bright82 for node003</td>
</tr>
<tr>
<td>node003</td>
<td>from_head_for_some_others</td>
<td>Test</td>
<td>76</td>
<td>0.08s</td>
<td>Sampled on node003 for node003</td>
</tr>
<tr>
<td>node004</td>
<td>from_head_for_some_others</td>
<td>Test</td>
<td>25</td>
<td>0.08s</td>
<td>Sampled on node003 for node004</td>
</tr>
<tr>
<td>node004</td>
<td>from_head_for_some_others</td>
<td>Test</td>
<td>88</td>
<td>0.151s</td>
<td>Sampled on bright82 for node004</td>
</tr>
</tbody>
</table>

```
[bright82->monitoring->setup]% nodes from_head_for_some_others
node003,bright82
```

The node node003 is now doing what the head node is, sampling the same targets, which is typically a bad idea. However the behavior is indeed as expected for this particular configuration.

Whether running a particular configuration is actually wise, is up to the administrator—but in any case the filter/multiplex system allows plenty of abuse of this kind.

### L.5 Not Just For Nodes

Nodes are what node execution filters and execution multiplexers run on. However, sometimes it is more convenient to execute based on category or other grouping types.

The possible types can be listed with tab-completion suggestions when adding a node execution filter or an execution multiplier:

```
[bright82->monitoring->setup[dmesg]->nodeexecutionfilter]% add<TAB><TAB>
```

| category lua node overlay resource type |

### L.6 Lua Node Execution Filters

Lua ([https://www.lua.org/](https://www.lua.org/)) is a lightweight scripting language embedded into CMDaemon. It allows more advanced node execution filters to be written using a Lua filter file.

**Example**

```
[bright82->monitoring->setup[<data producer>]->nodeexecutionfilters]% add lua lua-filter
[bright82->...nodeexecutionfilters+[lua-filter+]]% set code <Lua filter file name>
```

In the preceding example session, the name `lua-filter` is an arbitrary name, that is added to the object that is associated with the Lua filter file `<Lua filter file name>`.

The `self` Lua table is passed by CMDaemon, and contains the entity for which the filter is evaluated. Only devices have a `self` table that is not `nil`. All other entities have a `self` table that is `nil`.

For example, a filter can be created based on a regex match of the hostname:

**Example**

```
if self == nil then
  return false
else
  return self.hostname:match("node[0-9]+") ~= nil)
end
```

The Lua filter is evaluated for all nodes:

**Example**

© Bright Computing, Inc.
Development of Lua filters is best done outside of CMDaemon. Doing so requires the cluster administrator to create node environments to be evaluated by hand:

Example

[root@bright82 ~]# cat node001.lua
self = {}
self.hostname = "node001"
self.category = "default"

[root@bright82 ~]# cat bright82.lua
self = {}
self.hostname = "bright82"

Both nodes can then be run through the filter using the lua interpreter:

Example

[root@bright82 ~]# lua
Lua 5.1.4 Copyright (C) 1994-2015 Lua.org, PUC-Rio
> dofile('bright82.lua')
> print(dofile('filter.lua'))
false
> dofile('node001.lua')
> print(dofile('filter.lua'))
true

The Lua self exported by CMDaemon contains the following:

- **self**: The main table object, nil for non-devices.
- **self.hostname**: The hostname of the device.
- **self.partition**: The name of the partition to which the device belongs.
- **self.category**: The name of the category to which the compute node belongs.
- **self.nodegroups**: An array with the name of node groups the node belongs to.
- **self.mac**: The MAC address of the device.
- **self.ip**: The IP of the device, only available for non-node devices.
- **self.network**: The IP address of the device, only available for non-node devices.
- **self.interfaces**: The array of interface of a node.
- `self.interfaces[1].name`: The name of the first interface.
- `self.interfaces[1].ip`: The IP address of the first interface.
- `self.interfaces[1].network`: The name of the network of the first interface.
- `self.roles`: The array of roles of a node.
- `self.roles[1].name`: The name of the first role.
- `self.roles[1].type`: The type of the first role.
- `self.status.status`: The status of the device.
- `self.status.user_message`: The user message set for the device.
- `self.status.info_message`: The information message set for the device.
- `self.status.closed`: A boolean marking the node as closed.
- `self.status.restart_required`: A boolean marking the node needs to be rebooted.
- `self.status.healthcheck_failed`: A boolean marking at least one health check has returned `FAIL`.
- `self.status.healthcheck_unknown`: A boolean marking at least one health check has returned `UNKNOWN`.
- `self.status.state_flapping`: A boolean marking the node status transitioning often in a short time span
- `self.system.name`: The name of the system.
- `self.system.manufacturer`: The manufacturer of the system.
- `self.system.motherboard.name`: The name of the motherboard.
- `self.system.motherboard.manufacturer`: The manufacturer of the motherboard.
- `self.system.bios.version`: The BIOS version.
- `self.system.bios.vendor`: The BIOS vendor name.
- `self.system.bios.date`: The BIOS date.
- `self.system.os.name`: The OS name.
- `self.system.os.version`: The OS version.
- `self.system.os.flavor`: The OS flavor.

Using the `self` environment, complex filters can easily be created. For example, the following filter can be built to include only the nodes on the IB network, which also have a Slurm client role:

**Example**

```
[root@bright82 ~]# cat ib-slurm-filter.lua
if self == nil then
  return false
end

on_ib_network = false
```

© Bright Computing, Inc.
for index, interface in ipairs(self.interface) do
    on_ib_network = on_ib_network or (interface.network == "ibnet")
done

slurm_client = false
for index, role in ipairs(self.roles) do
    slurm_client = slurm_client or (role.name == "SlurmClient")
done

return on_ib_network and slurm_client

It is also possible to use external sources, like the file system, to determine the filter for a node.

Example

[root@bright82 ~]# cat file-check-filter.lua
if self == nil then
    return false
end

function file_exists(name)
    local f = io.open(name, "r")
    if f ~= nil then
        io.close(f)
        return true
    else
        return false
    end
end

return file_exists(string.format('/opt/filter/%s', self.hostname))

It is important to understand that this Lua script is evaluated for all nodes, on the active head node. The Lua script should therefore be fast, and return within a few milliseconds.