

Cloud Virtual Machine

Operations Guide

Product Documentation



Copyright Notice

©2013-2025 Tencent Cloud. All rights reserved.

Copyright in this document is exclusively owned by Tencent Cloud. You must not reproduce, modify, copy or distribute in any way, in whole or in part, the contents of this document without Tencent Cloud's the prior written consent.

Trademark Notice



All trademarks associated with Tencent Cloud and its services are owned by the Tencent corporate group, including its parent, subsidiaries and affiliated companies, as the case may be. Trademarks of third parties referred to in this document are owned by their respective proprietors.

Service Statement

This document is intended to provide users with general information about Tencent Cloud's products and services only and does not form part of Tencent Cloud's terms and conditions. Tencent Cloud's products or services are subject to change. Specific products and services and the standards applicable to them are exclusively provided for in Tencent Cloud's applicable terms and conditions.

Contents

Operations Guide

Initializing Data Disks

- Initializing Data Disks (Windows CVM)

- Formatting & Mounting Data Disks

- Reading/Writing EXT Data Disks after Reinstalling a Linux CVM to Windows CVM

- Read/Write of the Original NTFS Data Disk After the Operating System Is Changed from Windows to Linux

Environment Configurations

- Linux Power Management Configuration

- Modifying etc/hosts Configuration of Linux Instance

Installing Software

- Accelerating Software Package Downloading and Updating with Tencent Cloud Software Source

- Installing Software via Apt-get (Ubuntu)

- Installing Software via YUM (CentOS)

- Installing Software via zypper (SUSE)

- Switching CentOS 6 Source to YUM Repository

- Installing chronyd Service on CentOS 8

User Data

- Configuring Custom Data (Linux CVM)

- Setting Custom Data (Windows CVM)

Work with Operating Systems

System Activation

- Activating Windows with slmgr Command

- Windows Server System Activation

System Updates

System Shutdown

- Configuring High-performance Power Management

- Windows Recovery Mode

- Updating the Virtio ENI Driver

- Modifying SID

- Modifying VNC Resolution

- Ensuring Unique SIDs for CVMs Using Sysprep

- Use of the atop Monitoring Tools by Linux Instances

- Introduction to Linux Kernel Parameters

Others

- Booting into Linux Single User Mode

Using Rescue Mode

Setting Up Windows CVM for Multi-user Remote Login

Modifying the Default Remote Port of CVM

Getting Instance Login Logs

Operations Guide

Initializing Data Disks

Initializing Data Disks (Windows CVM)

Last updated : 2024-01-06 17:38:11

Scenario

After purchasing or reinstalling CVM, you need to partition and format the data disk. This document describes how to perform partitioning, formatting, and other initialization operations on a data disk on Windows CVM.

Notes

Formatting a data disk will erase all data. Please ensure that the data disk does not contain data, or important data has been backed up.

To avoid service exceptions, ensure before formatting that the CVM has stopped external services.

Directions

Please select the appropriate operation guide based on disk capacity:

If the disk capacity is less than 2 TB, please [initializing cloud disks](#).

If the disk capacity is greater than or equal to 2 TB, please [initializing cloud disks](#).

Formatting & Mounting Data Disks

Last updated : 2024-01-06 17:38:11

Overview

This document describes how to perform initialization operations such as formatting, partitioning, and creating a file system on a Linux CVM data disk.

Notes

Before formatting, ensure that no data is stored on the data disk and that important data has been backed up. After formatting, all data on the data disk will be cleared.

To prevent service exceptions, ensure that the CVM has stopped providing external services before formatting.

Directions

Follow the appropriate operation guide based on the disk capacity.

If the disk capacity is less than 2 TB, follow the instructions under [Initializing Cloud Disks \(Linux\)](#).

If the disk capacity is 2 TB or more, follow the instructions under [Initializing Cloud Disks \(Linux\)](#).

Reading/Writing EXT Data Disks after Reinstalling a Linux CVM to Windows CVM

Last updated : 2024-01-06 17:38:11

Scenario

Windows employs two major file systems, NTFS or FAT32, while EXT is the de factor file system for Linux. When an operating system is changed from Linux to Windows after reinstallation, the data disk remains in its original format. Therefore, the system might not be able to access data disk's file system. In these cases, you will need to use a format converter to read the data disk.

This document describes how to read a data disk when the operating system has been [reinstalled](#) from Linux to Windows.

Prerequisites

DiskInternals Linux Reader has been installed on the reinstalled Windows CVM.

Download DiskInternals Linux Reader: http://www.diskinternals.com/download/Linux_Reader.exe .

Suppose the data disk mounted to the Linux CVM before reinstallation has two partitions, vdb1 and vdb2, as shown below:

```
Disk /dev/vdb: 21.5 GB, 21474836480 bytes
16 heads, 63 sectors/track, 41610 cylinders
Units = cylinders of 1008 * 512 = 516096 bytes
Sector size (logical/physical): 512 bytes / 512 bytes
I/O size (minimum/optimal): 512 bytes / 512 bytes
Disk identifier: 0x29cc8ca2
```

Device	Boot	Start	End	Blocks	Id	System
/dev/vdb1		2000	41610	19963944	83	Linux
/dev/vdb2		1	1999	1007464+	83	Linux

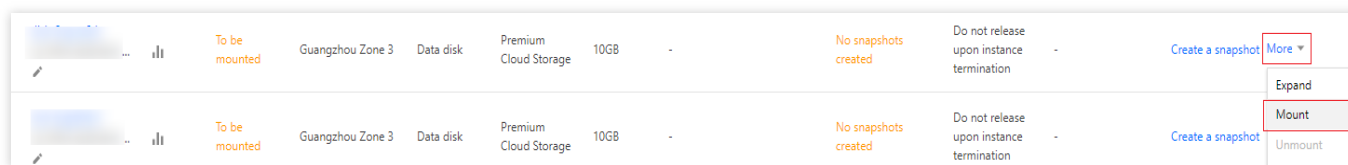
Directions

Mounting a data disk

Note:

If a data disk has been mounted, skip this step.

1. Log in to the [Tencent Cloud CVM Console](#).
2. Click **Cloud Block Storage** from the left sidebar to enter the Cloud Block Storage management page.
3. Locate the instance with the reinstalled system, and click **More > Mount** on the right as shown below:



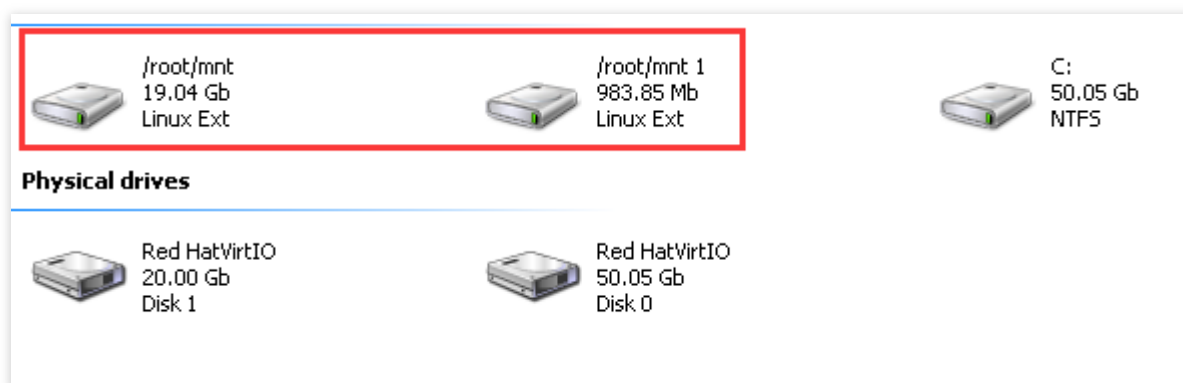
4. In the pop-up window, select the reinstalled Windows CVM and click **Submit**.

Viewing data disk information

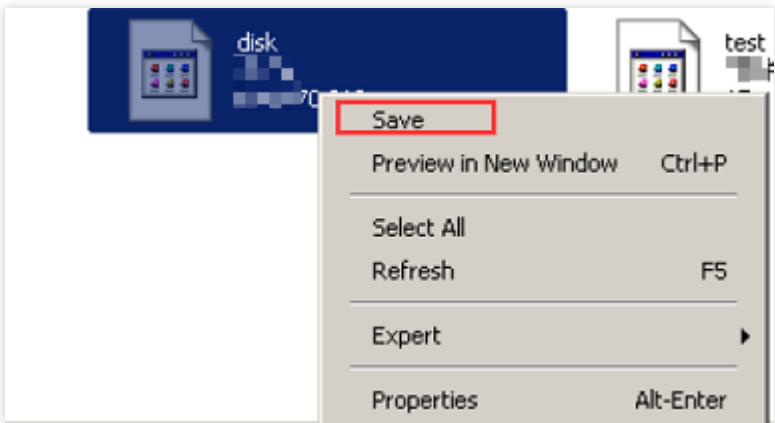
1. Run DiskInternals Linux Reader to view the information of newly mounted data disk. `/root/mnt` and `/root/mnt1` correspond to vdb1 and vdb2 respectively, which are the 2 data disk partitions on the Linux CVM before reinstallation as shown below:

Note:

Note that the Linux data disk is read-only at this time. To perform read and write operations on the data disk as you do on a Windows data disk, back up your needed files and re-format the disk into a standard Windows-supported file system. For more information, please see [Data Disk Partition and Formatting of Windows CVMs](#).



2. Double-click to enter `/root/mnt` directory, right-click the file you want to copy, and select **Save** as shown below:



Read/Write of the Original NTFS Data Disk After the Operating System Is Changed from Windows to Linux

Last updated : 2024-01-06 17:38:11

Overview

Windows file system typically uses NTFS or FAT32 format, while Linux file system often uses EXT series format. When the operating system of a CVM is reinstalled from Windows to Linux, the data disk of the CVM remains in the format of the original operating system. As a result, after the system reinstallation, the CVM might be unable to access the data disk file system. This document describes how to read a data disk in the original Windows system after the operating system is reinstalled from Windows to Linux.

Directions

Installing NTFS software on a Linux CVM

1. Log in to the Linux CVM after reinstallation.
2. Run the following command to install the ntfsprogs software to enable the Linux CVM to support access to the NTFS file system.

Note:

This document takes CentOS as an example. Different types of Linux systems have different installation commands. Please use the corresponding installation commands.

```
yum install -y ntfsprogs
```

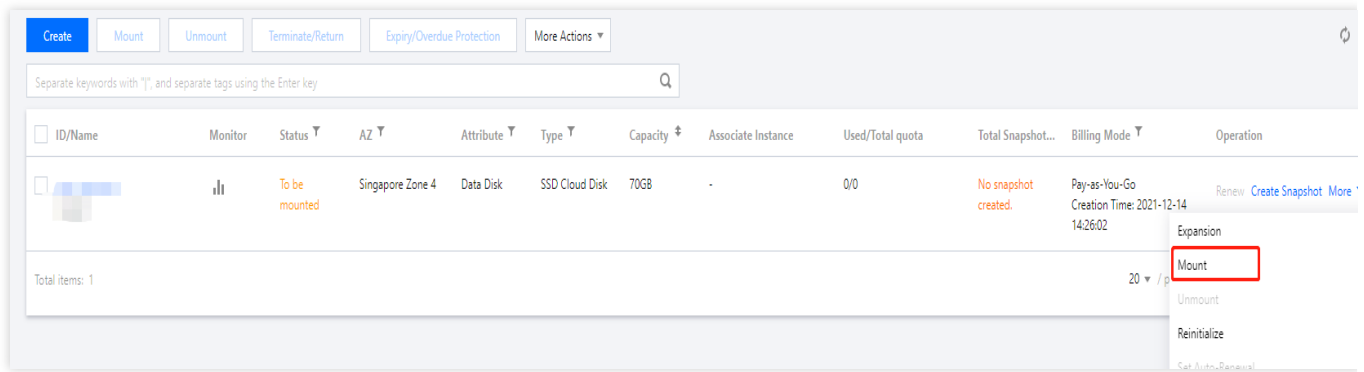
Attaching a data disk from the Windows CVM to the Linux CVM

Note:

If the data disk in your Windows CVM has been attached to the Linux CVM, skip this operation.

If you want to attach a new data disk to the reinstalled Linux CVM, please [Initialize Cloud Disks](#).

1. Log in to the [CVM console](#).
2. In the left sidebar, click [Cloud Block Storage](#) to enter the Cloud Block Storage management page.
3. Select the target Windows data disk. Click **More > Attach**.



4. In the pop-up window, choose the target Linux CVM, and click **OK**.
5. Log in to the Linux CVM to which the Windows data disk has been attached.
6. Run the following command to query the data disk attached from the Windows CVM.

```
parted -l
```

A message similar to the one below is returned:

```
Model: Virtio Block Device (virtblk)
Disk /dev/vdb: 53.7GB
Sector size (logical/physical): 512B/512B
Partition Table: gpt
Disk Flags:
Number  Start   End     Size    File system  Name
Flags
 1      17.4kB  134MB   134MB                   Microsoft reserved partition
msftres
 2      135MB   53.7GB  53.6GB   ntfs          Basic data partition
```

7. Run the following command to attach the data disk.

```
mount -t ntfs-3g Data disk path Mount point
```

For example, if you need to attach the data disk in `/dev/vdb2` to `/mnt`, run the following command:

```
mount -t ntfs-3g /dev/vdb2 /mnt
```

Since the file system is identifiable, Linux system can directly perform read and write operations on the attached data disk.

Environment Configurations

Linux Power Management Configuration

Last updated : 2024-01-06 17:38:11

Introduction

x86 machines use two power management methods, **APM** (Advanced Power Management) and **ACPI** (Advanced Configuration and Power Interface). ACPI is a power management standard jointly developed by Intel, Microsoft, and Toshiba, which provides a more flexible interface for computer and device management, whereas APM is the old power management standard.

Linux supports APM and ACPI, but the two standards cannot run simultaneously. Linux runs ACPI by default. Tencent Cloud also recommends ACPI.

If ACPI is not installed in a Linux system, the soft shutdown will fail. This document describes how to check whether ACPI has been installed and if not, how to install it.

Notes

For CoreOS, there is no need to install ACPI.

Directions

1. Execute the following command to see if ACPI has been installed.

```
ps -ef|grep -w "acpid"|grep -v "grep"
```

If there is no process running, it means ACPI has not been installed. Please go to the next step.

If there is a process running, it means ACPI has been installed.

2. Execute the following command to install ACPI.

For Ubuntu or Debian:

```
sudo apt-get install acpid
```

For Redhat or CentOS:

```
yum install acpid
```

For SUSE:

```
in apcid
```

Modifying etc/hosts Configuration of Linux Instance

Last updated : 2024-01-06 17:38:11

Overview

From March 1, 2018, the Linux public image provided by Tencent Cloud has the open-source tool Cloud-Init pre-installed, and all initialization operations on an instance are done via Cloud-Init, making operations inside the instance more transparent. For more information, see [Cloud-Init and Cloudbase-Init](#).

In **each launch**, Cloud-Init generates a new `/etc/hosts` file according to the `/etc/cloud/templates/hosts.${os_type}.tpl` template and overwrites the original `/etc/hosts` file of the instance involved. Hence, after you manually modify the internal `/etc/hosts` configuration of the instance and restart it, the `/etc/hosts` configuration goes back to the original default configuration.

Precautions

Tencent Cloud has fixed this problem for instances created by using a public image **after September 2018**, and the `/etc/hosts` configuration will not be overwritten.

For instance created **before September 2018**, follow the steps below for modification.

Directions

Solution 1

1. Log in to the Linux CVM.
2. Execute the following command to change the `- update_etc_hosts` in the `/etc/cloud/cloud.cfg` configuration file to `- ['update-etc-hosts', 'once-per-instance']` :

```
sed -i "/update_etc_hosts/c \\ - ['update_etc_hosts', 'once-per-instance']" /etc/cl
```

3. Execute the following command to create a `config_update_etc_hosts` file under the `/var/lib/cloud/instance/sem/` path:

```
touch /var/lib/cloud/instance/sem/config_update_etc_hosts
```

Solution 2

Note:

This solution takes the CentOS 7.2 operating system as an example.

Obtaining the `hosts` template file path

1. Log in to the Linux CVM.
2. Execute the following command to view the system `hosts` template file:

```
cat /etc/hosts
```

The `hosts` template file is as shown in the following figure:

```
[root@UM_2_9_centos ~]# cat /etc/hosts
# Your system has configured 'manage_etc_hosts' as True.
# As a result, if you wish for changes to this file to persist
# then you will need to either
# a.) make changes to the master file in /etc/cloud/templates/hosts.redhat.tpl
# b.) change or remove the value of 'manage_etc_hosts' in
#    /etc/cloud/cloud.cfg or cloud-config from user-data
#
# The following lines are desirable for IPv4 capable hosts
127.0.0.1 UM_2_9_centos UM_2_9_centos
127.0.0.1 localhost.localdomain localhost
127.0.0.1 localhost4.localdomain4 localhost4

# The following lines are desirable for IPv6 capable hosts
::1 UM_2_9_centos UM_2_9_centos
::1 localhost.localdomain localhost
::1 localhost6.localdomain6 localhost6
[root@UM_2_9_centos ~]# _
```

Modifying the `hosts` template file

Note:

Taking adding `127.0.0.1 test test` as an example, you can modify the `hosts` template and `/etc/hosts` file as needed.

1. Execute the following command to modify the `hosts` template file:

```
vim /etc/cloud/templates/hosts.redhat.tpl
```

2. Press `i` to switch to the editing mode.
3. Add the following content to the end of the file:

```
127.0.0.1 test test
```

4. Press **Esc** and enter `:wq` to save and close the file.

Modifying the `/etc/hosts` file

1. Execute the following command to modify the `/etc/hosts` file:

```
vim /etc/hosts
```

2. Press **i** to switch to the editing mode.
3. Add the following content to the end of the file:

```
127.0.0.1 test test
```

4. Press **Esc** and enter **:wq** to save and close the file.

Installing Software Accelerating Software Package Downloading and Updating with Tencent Cloud Software Source

Last updated : 2024-01-06 17:38:11

Overview

To solve the problem of slow access to official sources when installing dependencies, Tencent Cloud has set up a cache service for some software. You can accelerate the installation of dependencies by using the Tencent Cloud software repository, which currently supports public network access and private network access.

Public network access address: `http://mirrors.tencent.com`

Private network access address: `http://mirrors.tencentyun.com/`

Note:

This document takes the public network access address of the Tencent Cloud software repository as an example to introduce how to use the software sources in the Tencent Cloud software repository in CVM. If you access the repository using a private network, please replace the public network access address **with the private network access address**.

The source address used in this document is for reference only. Please obtain the latest address from the **Tencent Cloud software repository**.

Notes

The Tencent Cloud software repository updates software sources from the official website of each software source once per day.

Prerequisites

Log in to the CVM.

Directions

Accelerating pip using the Tencent Cloud image source

Note:

Before using the Tencent Cloud image source, please confirm your CVM has Python installed.

Use the software source path temporarily

Execute the following command to install pip using Tencent Cloud PyPI.

```
pip install pip -i the directory where PyPI is located in
```

For example, if you want to install the 17monip package, and the PyPI you need to use is in the

`http://mirrors.tencent.com/pypi/simple` directory, run the following command:

```
pip install 17monip -i http://mirrors.tencent.com/pypi/simple --trusted-host mirrors.tencent.com
```

Set the default software source path

Execute the following command to modify the `index-url` parameter in the `~/.pip/pip.conf` file to the source path of the Tencent Cloud software repository.

```
[global]
index-url = the directory where PyPI is located in
trusted-host = public network/private network access address
```

For example, if the PyPI you need to use is in the `http://mirrors.tencent.com/pypi/simple` directory, execute the following command:

```
[global]
index-url = http://mirrors.tencent.com/pypi/simple
trusted-host = mirrors.tencent.com
```

Accelerating Maven using the Tencent Cloud image source

Note:

Before using the Tencent Cloud image source, please confirm your CVM has JDK and Maven installed.

1. Open the `settings.xml` configuration file of Maven.
2. Find the `<mirrors> ... </mirrors>` code block and configure the following content into it.

```
<mirror>
  <id>nexus-tencentyun</id>
  <mirrorOf>*</mirrorOf>
  <name>Nexus tencentyun</name>
  <url>http://mirrors.tencent.com/nexus/repository/maven-public/</url>
</mirror>
```

Accelerating NPM using the Tencent Cloud image source

Note:

Before using the Tencent Cloud image source, please confirm your CVM has Node.js and NPM installed. Execute the following command to install NPM using the Tencent Cloud NPM.

```
npm config set registry http://mirrors.tencent.com/npm/
```

Accelerating Docker using the Tencent Cloud image source

Using Tencent Cloud Docker on the TKE cluster

No manual configuration is required. When the CVM in the Tencent Kubernetes Engine (TKE) cluster creates a node, Docker will be installed automatically and configured with the Tencent Cloud private network image.

Using Tencent Cloud Docker on CVM

Note:

Before using the Tencent Cloud Docker, please confirm your CVM has Docker installed.

Only Docker 1.3.2 or later versions support the Docker Hub Mirror mechanism. If you have not installed Docker 1.3.2 or later versions, or if the installed version is too old, please install or upgrade it first.

Choose different operation steps based on the operating system of the CVM.

The following steps are for Ubuntu 14.04, Debian, CentOS 6, Fedora, openSUSE, and other operating systems. The specific steps for other versions of operating systems may vary:

1.1 Execute the following command to open the `/etc/default/docker` configuration file.

```
vim /etc/default/docker
```

1.2 Press **i** to switch to the editing mode, enter the following content, and save.

```
DOCKER_OPTS="--registry-mirror=https://mirror.ccs.tencentyun.com"
```

For Centos 7:

1.1 Execute the following command to open the `/etc/docker/daemon.json` configuration file.

```
vim /etc/docker/daemon.json
```

1.2 Press **i** to switch to the editing mode, enter the following content, and save.

```
{
  "registry-mirrors": [
    "https://mirror.ccs.tencentyun.com"
  ]
}
```

For Windows with Boot2Docker installed:

1.1 Access the Boot2Docker Start Shell and execute the following command:

```
sudo su echo "EXTRA_ARGS=\\\"-registry-mirror=https://mirror.ccs.tencentyun.com\\\" >> /var/lib/boot2docker/profile
exit
```

1.2 Restart Boot2Docker.

Accelerating MariaDB using the Tencent Cloud image

Note:

The following steps take CentOS 7 as an example. Specific steps vary by operating system.

1. Execute the following command to create the `MariaDB.repo` file under `/etc/yum.repos.d/`.

```
vi /etc/yum.repos.d/MariaDB.repo
```

2. Press `i` to switch to the editing mode, enter the following content, and save.

```
# MariaDB 10.2 CentOS7-amd64
[mariadb]
name = MariaDB
baseurl = http://mirrors.tencent.com/mariadb/yum/10.2/centos7-amd64/
gpgkey = http://mirrors.tencent.com/mariadb/yum/RPM-GPG-KEY-MariaDB
gpgcheck=1
```

3. Execute the following command to empty the YUM cache.

```
yum clean all
```

4. Execute the following command to install MariaDB.

```
yum install MariaDB-client MariaDB-server
```

Accelerating MongoDB using the Tencent Cloud image

Note:

The following steps take MongoDB 4.0 as an example. If you need to install another version, please change the version number in the mirror path.

Using Tencent Cloud MongoDB on CVMs with CentOS or Redhat systems

1. Execute the following command to create the `mongodb.repo` file under `/etc/yum.repos.d/`.

```
vi /etc/yum.repos.d/mongodb.repo
```

2. Press `i` to switch to the editing mode, enter the following content, and save.

```
[mongodb-org-4.0]
name=MongoDB Repository
baseurl=http://mirrors.tencent.com/mongodb/yum/el7-4.0
gpgcheck=0
enabled=1
```

3. Execute the following command to install MongoDB.

```
yum install -y mongodb-org
```

Using Tencent Cloud MongoDB on CVMs with the Debian system

1. Based on the different Debian versions, execute the following command to import the MongoDB GPG public key.

```
sudo apt-key adv --keyserver hkp://keyserver.ubuntu.com:80 --recv
68818C72E52529D4
```

2. Execute the following command to configure the `mirror` path.

```
#Debian 8
echo "deb http://mirrors.tencent.com/mongodb/apt/debian jessie/mongodb-org/4.0
main" | sudo tee /etc/apt/sources.list.d/mongodb-org-4.0.list
#Debian 9
echo "deb http://mirrors.tencent.com/mongodb/apt/debian stretch/mongodb-org/4.0
main" | sudo tee /etc/apt/sources.list.d/mongodb-org-4.0.list
```

3. Run the following command to clear the cache.

```
sudo apt-get clean all
```

4. Run the following command to update the software package list.

```
sudo apt-get update
```

5. Execute the following command to install MongoDB.

```
sudo apt-get install -y mongodb-org
```

Using Tencent Cloud MongoDB on CVMs with the Ubuntu system

1. Execute the following command to import the MongoDB GPG public key.

```
sudo apt-key adv --keyserver hkp://keyserver.ubuntu.com:80 --recv
68818C72E52529D4
```

2. Execute the following command to configure the `mirror` path.

```
#Ubuntu 14.04
echo "deb [ arch=amd64 ] http://mirrors.tencent.com/mongodb/apt/ubuntu
trusty/mongodb-org/4.0 multiverse" | sudo tee /etc/apt/sources.list.d/mongodb-
org-4.0.list
#Ubuntu 16.04
echo "deb [ arch=amd64 ] http://mirrors.tencent.com/mongodb/apt/ubuntu
xenial/mongodb-org/4.0 multiverse" | sudo tee /etc/apt/sources.list.d/mongodb-
org-4.0.list
#Ubuntu 18.04
echo "deb [ arch=amd64 ] http://mirrors.tencent.com/mongodb/apt/ubuntu
bionic/mongodb-org/4.0 multiverse" | sudo tee /etc/apt/sources.list.d/mongodb-
org-4.0.list
```

3. Run the following command to clear the cache.

```
sudo apt-get clean all
```

4. Run the following command to update the software package list.

```
sudo apt-get update
```

5. Execute the following command to install MongoDB.

```
sudo apt-get install -y mongodb-org
```

Accelerating Rubygems using the Tencent Cloud image source

Note:

Before using the Tencent Cloud image source, please confirm your CVM has Ruby installed.

Execute the following command to modify the RubyGems source address.

```
gem source -r https://rubygems.org/
gem source -a http://mirrors.tencent.com/rubygems/
```

Installing Software via Apt-get (Ubuntu)

Last updated : 2024-01-06 17:38:11

Introduction

Advanced Package Tool (APT) is a free software user interface that works with core libraries to handle the installation and removal of software on various Linux distributions. APT offers a centralized interface for software management and a better experience than having to download and install software one by one. Tencent Cloud hosts APT sources so you can install software without adding sources.

Prerequisites

Log in to a Linux CVM instance that runs Ubuntu.

Directions

Note:

In the following, Nginx is used as an example.

Listing available software

Run the following command to list available software:

```
sudo apt-cache search all
```

Installing software

Run the following command to install Nginx:

```
sudo apt-get install nginx
```

Make sure this is the software you want to install and enter **Y** to approve the installation. Wait until the software installation is complete, as shown in the figure below:

```
ubuntu@VM-179-94-ubuntu:~$ sudo apt-get install nginx
Reading package lists... Done
Building dependency tree
Reading state information... Done
The following additional packages will be installed:
  fontconfig-config fonts-dejavu-core libfontconfig1 libgd3 libjpeg-turbo8 libjpeg8 libtiff5 libvpx3 libxpm4 libxslt1.1 nginx-common nginx-core
Suggested packages:
  libgd-tools fcgiwrap nginx-doc ssl-cert
The following NEW packages will be installed:
  fontconfig-config fonts-dejavu-core libfontconfig1 libgd3 libjpeg-turbo8 libjpeg8 libtiff5 libvpx3 libxpm4 libxslt1.1 nginx nginx-common nginx-
0 upgraded, 14 newly installed, 0 to remove and 186 not upgraded.
Need to get 3,000 kB of archives.
After this operation, 9,782 kB of additional disk space will be used.
Do you want to continue? [Y/n]
```

Querying installed software

You can run different commands to query installed software.

Run the following command to query the directory of the software package and all the files in the software package.

```
sudo dpkg -L software_name
```

Run the following command to query the version information of the software package.

```
sudo dpkg -l software_name
```

View information on the installed Nginx, as shown in the figure below:

```
ubuntu@VM-179-94-ubuntu:~$ sudo dpkg -L nginx
./
/usr
/usr/share
/usr/share/doc
/usr/share/doc/nginx
/usr/share/doc/nginx/copyright
/usr/share/doc/nginx/changelog.Debian.gz
ubuntu@VM-179-94-ubuntu:~$ sudo dpkg -l nginx
Desired=Unknown/Install/Remove/Purge/Hold
| Status=Not/Inst/Conf-files/Unpacked/halF-conf/Half-inst/trig-aWait/Trig-pend
|/ Err?=(none)/Reinst-required (Status,Err: uppercase=bad)
||/ Name                      Version                      Architecture                Description
+++=====
ii  nginx                        1.10.3-0ubuntu0.16.04. all    small, powerful, scalable web/proxy server
```


Installing Software via YUM (CentOS)

Last updated : 2024-01-06 17:38:11

Overview

YUM (Yellowdog Updater, Modified) allows you to easily download and install software, and simplifies your installations on a CVM instance, saving you time and efforts. With it, you only need to run the `yum` command to install software in the CentOS environment. Tencent Cloud provides a YUM repository so you can directly install software packages without adding sources.

Directions

Installing software

Log in to your CVM instance with the root account and run the following commands to install software according to the operating system of your CVM.

CentOS 8 or later versions

1. Run the following command to install software.

```
dnf install [software name]
```

The system will automatically search for the relevant software package and dependencies, and ask for your confirmation.

For example, after you run the `dnf install php` command to install PHP, the following prompt will appear:

```
[root@VM-16-67-centos ~]# dnf install php
Last metadata expiration check: 0:07:41 ago on Fri 20 Nov 2020 04:38:02 PM CST.
Dependencies resolved.
=====
Package                Arch          Version                               Repository    Size
=====
Installing:
php                    x86_64        7.2.24-1.module_el8.2.0+313+b04d0a66 AppStream     1
Installing dependencies:
apr                    x86_64        1.6.3-9.el8                          AppStream     1
apr-util              x86_64        1.6.1-6.el8                          AppStream     1
httpd                 x86_64        2.4.37-21.module_el8.2.0+494+1df74eae AppStream     1
httpd-filesystem      noarch        2.4.37-21.module_el8.2.0+494+1df74eae AppStream
httpd-tools           x86_64        2.4.37-21.module_el8.2.0+494+1df74eae AppStream     1
mod_http2             x86_64        1.11.3-3.module_el8.2.0+486+c01050f0.1 AppStream     1
nginx-filesystem      noarch        1:1.14.1-9.module_el8.0.0+184+e34fea82 AppStream
php-cli               x86_64        7.2.24-1.module_el8.2.0+313+b04d0a66 AppStream     3
php-common            x86_64        7.2.24-1.module_el8.2.0+313+b04d0a66 AppStream     6
centos-logos-httpd    noarch        80.5-2.el8                           BaseOS
Installing weak dependencies:
apr-util-bdb          x86_64        1.6.1-6.el8                          AppStream
apr-util-openssl      x86_64        1.6.1-6.el8                          AppStream
php-fpm               x86_64        7.2.24-1.module_el8.2.0+313+b04d0a66 AppStream     1
Enabling module streams:
httpd                 2.4
nginx                 1.14
php                   7.2

Transaction Summary
=====
Install 14 Packages

Total download size: 9.2 M
Installed size: 31 M
Is this ok [y/N]: 
```

2. Confirm that the software package is correct. Enter `y` and press **Enter** to start the installation.

The `Complete` prompt indicates the installation is completed.

CentOS 7 or earlier versions

1. Run the following command to install software.

Note:

Starting from CentOS 7, MariaDB has become the default database in the YUM repository. If you are using CentOS 7 or later versions, MySQL installed using `yum` will be unusable. You can use the fully compatible MariaDB, or [click here](#) to learn about how to install an older version of MySQL.

```
yum install [software name]
```

The system will automatically search for the relevant software package and dependencies, and ask for your confirmation.

For example, after you run the `yum install PHP` command to install PHP, the following prompt will appear:

```

=====
Package                Arch                Version              Repository            S
=====
Installing:
php                    x86_64              5.4.16-46.el7        os                    1.
Installing for dependencies:
apr                    x86_64              1.4.8-3.el7_4.1      os                    10
apr-util               x86_64              1.5.2-6.el7          os                    9
httpd                  x86_64              2.4.6-89.el7.centos  updates              2.
httpd-tools            x86_64              2.4.6-89.el7.centos  updates              9
libzip                 x86_64              0.10.1-8.el7         os                    4
mailcap                noarch              2.1.41-2.el7         os                    3
php-cli                x86_64              5.4.16-46.el7        os                    2.
php-common             x86_64              5.4.16-46.el7        os                    56

Transaction Summary
=====
Install 1 Package (+8 Dependent packages)

Total download size: 7.7 M
Installed size: 27 M
Is this ok [y/d/N]:

```

2. Confirm that the software package is correct. Enter `y` and press **Enter** to start the installation.

The `Complete` prompt indicates the installation is completed.

Viewing installed software information

After software installation is completed, you can run different commands to view related information.

View the installation directory of a software package

```
rpm -ql [software name]
```

For example, run `rpm -ql php` to view the installation directory of PHP, as shown in the figure below:

```

[root@VM_5_10_centos ~]# rpm -ql php
/etc/httpd/conf.d/php.conf
/etc/httpd/conf.modules.d/10-php.conf
/usr/lib64/httpd/modules/libphp5.so
/usr/share/httpd/icons/php.gif
/var/lib/php/session
[root@VM_5_10_centos ~]#

```

View the version of the software package

```
rpm -q
```

For example, run `rpm -q php` to view the version of PHP, as shown in the figure below:

```
[root@VM_5_10_centos ~]# rpm -q php
php-5.4.16-46.el7.x86_64
[root@VM_5_10_centos ~]#
```

Installing Software via zypper (SUSE)

Last updated : 2024-01-06 17:38:11

Make sure that you have followed the steps in [Installing Software via YAST in SUSE Environment](#) install the necessary software.

1. Configuration of nginx

1. Start nginx service

Start the nginx with the following command:

```
service nginx restart
```

2. Test whether nginx service is working properly

Test with the following command:

```
wget http://127.0.0.1
```

If the result is as shown below and displays "'index.html' saved" at the end, it means the nginx service is working properly.

```
--2013-02-20 17:07:26-- http://127.0.0.1/  
Connecting to 127.0.0.1:80... connected.  
HTTP request sent, awaiting response... 200 OK  
Length: 151 [text/html]  
Saving to: 'index.html'  
100%  
[=====]  
=====>] 151 --.-K/s in 0s  
2013-02-20 17:07:26 (37.9 MB/s) - 'index.html' saved [151/151]
```

3. In the browser, visit the Public IP of CentOS CVM to check if the nginx service is working properly.

The appearance of the following page indicates that nginx has been installed and configured successfully.

2. Configuration of PHP

1. Create a new configuration file php-fpm.conf with the following command:

```
vim /etc/php5/fpm/php-fpm.conf
```

Write the following:

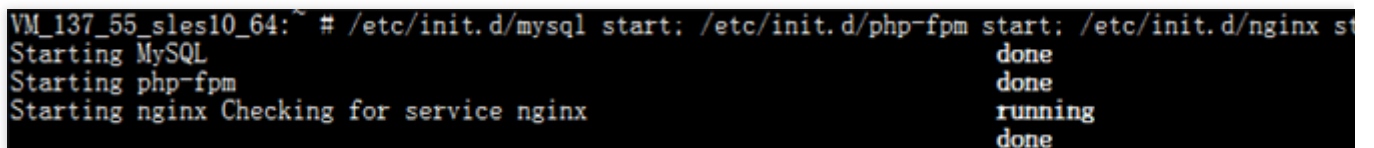
```
[global]
error_log = /var/log/php-fpm.log
[www]
user = nobody
group = nobody
listen = 127.0.0.1:9000
pm = dynamic
pm.max_children = 5
pm.start_servers = 2
pm.min_spare_servers = 1
pm.max_spare_servers = 3
```

3. Start services

Start all services with the following commands:

```
/etc/init.d/mysql start; /etc/init.d/php-fpm start; /etc/init.d/nginx start
```

Example:



A terminal window showing the execution of service startup commands. The prompt is 'VM_137_55_sles10_64:~ #'. The commands entered are '/etc/init.d/mysql start;', '/etc/init.d/php-fpm start;', and '/etc/init.d/nginx start'. The output shows 'Starting MySQL' followed by 'done', 'Starting php-fpm' followed by 'done', and 'Starting nginx Checking for service nginx' followed by 'running' and 'done'.

4. Environment configuration validation

Create index.php under a web directory using the following command:

```
vim /usr/share/nginx/html/index.php
```

Write the following:

```
<?php
echo "<title>Test Page</title>";
echo "hello world";
?>
```

In the browser, visit the Public IP of SUSE CVM to check whether the environment configuration is successful. If the webpage shows "hello world", it means the configuration is successful.

Switching CentOS 6 Source to YUM Repository

Last updated : 2024-01-06 17:38:11

Overview

The CentOS 6 is an end-of-life (EOL) operating system since November 30, 2020, and no longer maintained by the Linux community. Therefore, the CentOS 6 source is unavailable in `http://mirror.centos.org/centos-6/`, a third-party image site, or Tencent Cloud at `http://mirrors.tencent.com/` and `http://mirrors.tencentyun.com/`. If you continue to use the default CentOS 6 source configured in Tencent Cloud, an error may occur.

Note:

Upgrading your operating system to CentOS 7 or a later version is recommended. If you still need to use the CentOS 6 dependencies during the transition period, switch the CentOS 6 source as instructed below.

Directions

1. Log in to the Linux instance. [Logging in via Web Shell is recommended.](#) You can also use other login methods as required.

[Logging In to Linux Instances \(Remote Login\)](#)

[Logging In to Linux Instance \(SSH Key\)](#)

2. Run the following command to check the CentOS version of the current operating system.

```
cat /etc/centos-release
```

If the result as shown in the following figure is returned, the operating system version is CentOS 6.9.

```
[root@VM-2-14-centos ~]# cat /etc/centos-release
CentOS release 6.9 (Final)
```

3. Run the following command to edit the `CentOS-Base.repo` file.

```
vim /etc/yum.repos.d/CentOS-Base.repo
```

4. Press `i` to enter edit mode and modify `baseurl` according to the CentOS version and network environment.

Note:

See [Private Network Access](#) and [Internet Access](#) to determine the required source:

Source for private network access: `http://mirrors.tencentyun.com/centos-vault/6.x/`

Source for public network access: `http://mirrors.tencent.com/centos-vault/6.x/`

This document uses CentOS 6.9 that requires a private network access as an example. The modified `CentOS-Base.repo` file is as follows:

```
[extras]
gpgcheck=1
gpgkey=http://mirrors.tencentyun.com/centos/RPM-GPG-KEY-CentOS-6
enabled=1
baseurl=http://mirrors.tencentyun.com/centos-vault/6.9/extras/$basearch/
name=Qcloud centos extras - $basearch
[os]
gpgcheck=1
gpgkey=http://mirrors.tencentyun.com/centos/RPM-GPG-KEY-CentOS-6
enabled=1
baseurl=http://mirrors.tencentyun.com/centos-vault/6.9/os/$basearch/
name=Qcloud centos os - $basearch
[updates]
gpgcheck=1
gpgkey=http://mirrors.tencentyun.com/centos/RPM-GPG-KEY-CentOS-6
enabled=1
baseurl=http://mirrors.tencentyun.com/centos-vault/6.9/updates/$basearch/
name=Qcloud centos updates - $basearch
```

You can configure as follows:

```
[extras]
gpgcheck=1
gpgkey=http://mirrors.tencentyun.com/centos/RPM-GPG-KEY-CentOS-6
enabled=1
baseurl=http://mirrors.tencentyun.com/centos-vault/6.9/extras/$basearch/
name=Qcloud centos extras - $basearch
[os]
gpgcheck=1
gpgkey=http://mirrors.tencentyun.com/centos/RPM-GPG-KEY-CentOS-6
enabled=1
baseurl=http://mirrors.tencentyun.com/centos-vault/6.9/os/$basearch/
name=Qcloud centos os - $basearch
[updates]
gpgcheck=1
gpgkey=http://mirrors.tencentyun.com/centos/RPM-GPG-KEY-CentOS-6
```



```
enabled=1
baseurl=http://mirrors.tencentyun.com/centos-vault/6.9/updates/$basearch/
name=Qcloud centos updates - $basearch
```

5. Press **ESC**, enter **:wq**, and press **Enter** to save the modification.

6. Run the following command to modify the `CentOS-Epel.repo` file.

```
vim /etc/yum.repos.d/CentOS-Epel.repo
```

7. Press **i** to enter edit mode and modify `baseurl` based on the network environment.

In this example, change `baseurl=http://mirrors.tencentyun.com/epel/$releasever/$basearch/` to `baseurl=http://mirrors.tencentyun.com/epel-archive/6/$basearch/`. The result is as follows:

```
[epel]
name=epel for redhat/centos $releasever - $basearch
failovermethod=priority
gpgcheck=1
gpgkey=http://mirrors.tencentyun.com/epel/RPM-GPG-KEY-EPEL-6
enabled=1
baseurl=http://mirrors.tencentyun.com/epel-archive/6/$basearch/
```

You can configure as follows:

```
[epel]
name=epel for redhat/centos $releasever - $basearch
failovermethod=priority
gpgcheck=1
gpgkey=http://mirrors.tencentyun.com/epel/RPM-GPG-KEY-EPEL-6
enabled=1
baseurl=http://mirrors.tencentyun.com/epel-archive/6/$basearch/
```

8. Press **ESC**, enter **:wq**, and press **Enter** to save the modification.

9. Now you can use the `yum install` command to install the required software.

Installing chronyd Service on CentOS 8

Last updated : 2024-01-06 17:38:11

Overview

Currently, native CentOS 8 does not support setting the NTP service. Instead, you need to use chronyd to deliver accurate time. This document describes how to install and configure chronyd on a CentOS 8-based Tencent Cloud CVM instance.

Directions

Installing and configuring the chronyd service

1. [Log in to a Linux instance using standard login method.](#)
2. Run the following command to install the chronyd service.

```
yum -y install chrony
```

3. Run the following command to modify the `chrony.conf` configuration file.

```
vim /etc/chrony.conf
```

4. Press `i` to enter the edit mode, and append the following content to the next line of `#log measurements statistics tracking`.

```
server time1.tencentyun.com iburst
server time2.tencentyun.com iburst
server time3.tencentyun.com iburst
server time4.tencentyun.com iburst
server time5.tencentyun.com iburst
```

The result should be as follows:

```
# Get TAI-UTC offset and leap seconds from the system tz data
leapsectz right/UTC

# Specify directory for log files.
logdir /var/log/chrony

# Select which information is logged.
#log measurements statistics tracking
server time1.tencentyun.com iburst
server time2.tencentyun.com iburst
server time3.tencentyun.com iburst
server time4.tencentyun.com iburst
server time5.tencentyun.com iburst
```

5. Press **Esc** and enter **:wq** to save and close the file.

6. Run the following commands in sequence to enable chronyd autostart and restart it.

```
systemctl restart chronyd
```

```
systemctl enable chronyd
```

Verifying the configurations

1. Run the following command to check whether the clock is synced.

```
date
```

2. Run the following command to check the clock source status.

```
chronyc sourcestats -v
```

If a result similar to the following is returned, the configuration was successful.

```
[root@VM-64-40-centos ~]# chronyc sourcestats -v
210 Number of sources = 5
```

Name/IP Address	NP .- Number of sample points in measurement set.	NR ./.- Number of residual runs with same sign.	Span ./.- Length of measurement set (time).	Frequency ./.- Est. clock freq error (ppm).	Freq Skew ./.- Est. error in freq.	Offset .- Est. offset from reference. On the -. samples.	Std Dev
169.254.0.79	32	19	103m	-0.007	0.109	+868us	263
169.254.0.80	33	17	104m	+0.030	0.058	+126us	163
169.254.0.81	33	22	104m	-0.137	0.124	-1215us	291
169.254.0.82	30	16	97m	-0.015	0.126	+562us	284
169.254.0.83	33	16	104m	+0.012	0.079	-233us	177

Appendix

Commands

Command	Description
<code>chronyc sources -v</code>	Check the clock source.
<code>chronyc sourcestats -v</code>	Check the clock source status.
<code>timedatectl set-local-rtc 1</code>	Set the real time clock. The default format is UTC.
<code>timedatectl set-ntp yes</code>	Enable the NTP service for synchronization.
<code>chronyc tracking</code>	Calibrate the NTP server.
<code>chronyc -a makestep</code>	Force synchronization of the system clock.

User Data

Configuring Custom Data (Linux CVM)

Last updated : 2024-01-06 17:38:11

Overview

When creating a CVM instance, you can configure an instance by specifying **custom data**. During the **first launch** of the CVM, the custom data will be passed into the CVM in text format and be executed. If you purchase multiple CVM instances at a time, the custom data text will be executed on all CVM instances during their first launch.

This document describes how to transfer a shell script when a CVM instance for Linux is started for the first time.

Notes

Linux operating systems that support custom data include:

64-bit OSs: CentOS 6.8 and later, Ubuntu Server 14.04.1 LTS and later, and openSUSE 42.3 x86

32-bit OSs: CentOS 6.8 and later

A command can be executed by passing text only when a CVM instance is launched for the first time.

The text to be transferred must be Base64-encoded. **Perform encoding in a Linux environment to avoid format incompatibility.**

Execute the user input as `root`. Therefore, the `sudo` command is not required in the script. The `root` user can access all the files you created. If you need to grant other users with the access permission, modify the permission in the script.

During launch, executing specified tasks in custom data will increase the amount of time it takes to launch the CVM.

We recommend that you wait for a few minutes, and after the tasks are completed, test whether the tasks have been successfully executed.

In this sample, the shell script must start with `#!` and the path directing to the interpreter of the script to be read (generally starting with `/bin/bash`).

Directions

Writing a shell script

1. Run the following command to create a shell script named "script_text.sh".

```
vi script_text.sh
```

2. Press **I** to switch to the editing mode, refer to the following content, write it into the file, and save the "script_text.sh" script.

```
#!/bin/bash
echo "Hello Tencent Cloud."
```

Note:

The shell script must start with `#!` and the path directing to the interpreter of the script to be read (generally starting with `/bin/bash`). For more information on the shell script, see [BASH Programming - Introduction HOW-TO](#).

Base64-encoding the script file

1. Run the following command to Base64-encode the "script_text.sh" script.

```
# Base64-encode the script
base64 script_text.sh
```

The following information is returned:

```
# Result returned after the encoding
IyEvYmluL2Jhc2gKZWNoYAiSGVsbG8gVG VuY2VudCBDbG91ZC4iCg==
```

2. Run the following command to verify the result returned after the script is Base64-encoded.

```
# Base64-decode the returned result to verify the command
echo "IyEvYmluL2Jhc2gKZWNoYAiSGVsbG8gVG VuY2VudCBDbG91ZC4iCg==" | base64 -d
```

Passing text

We provide multiple methods to launch an instance, and here we introduce two of them. Choose a method according to your requirements:

Using the official website or the console

Using API

1. Purchase an instance as instructed in [Creating Instances via CVM Purchase Page](#) and click **Advanced Settings** in "2. Complete configuration".

1. Select a model

2. Complete Configuration

3. Confirm Configuration

Instance Name

(Optional) Defaults to "unnamed" if it's

Supports batch sequential naming or pattern string-based naming. You can enter 60 character(s)

Login Methods

Custom Password

SSH Key Pair

Random Password

Username

root

SSH Key

Select a key

Operation Guide

If no suitable key is found, you can

Create Now

Security Service

☒ Enable for Free

Install the component to activate Anti-DDoS service and host security service for free

Details

Cloud Monitoring

☒ Enable for Free

Activate cloud monitoring

Details

Scheduled Termination

☐ Enable Scheduled Termination

Enable it to terminate CVM at a specified time.

Advanced Settings

2. In the text box in **Advanced Settings**, enter the encoded result returned in the [Base64-encoded script](#).

For example, the encoded result of the `script_text` script with Base64 is

```
IyEvYmluL2Jhc2gKZWNoYAiSGVsbG8gVGVuY2VudCBDbG91ZC4iCg==
```

Advanced Settings

Hostname

(Optional) Computer name in the operation system. Supports batch sequential naming or pattern string-based naming. 2-60 characters, including uppercase and lowercase letters, numbers, hyphens "-" and dots ".". It supports {R:number} format, and does not allow colons ":" and braces "{}" are not allowed. Hyphens "-" and dots "." cannot be used consecutively, and cannot be placed at the beginning or end. / number-only password is not allowed.

Placement Group

☐ Add the instance to a placement group

Tag

Tag key	Tag value	Operation
Select a tag key	Select a tag value	Delete

Add

If the existing tags or tag values are not suitable, you can go to the console to [Create](#)

Custom Data

(Optional) It's used to configure instances while launching. Support Shell format. The original data cannot exceed 16KB. Shell script must start with "#" or "!" and the path to the interpreter reading the script (usually /bin/bash).

☒ The content above is Base64 encoded.

3. Follow the prompts on the interface to complete CVM creation.

Note:

CVM will run the script through the open-source software cloud-init. For more information on cloud-init, visit the [cloud-init website](#).

When creating a CVM instance through APIs, you can pass the text by assigning the value of the encoded result returned in the [Base64-encoded script](#) to the `UserData` parameter of the `RunInstances` API.

The following is an sample CVM creation request with UserData:

```
https://cvm.tencentcloudapi.com/?Action=RunInstances
&Version=2017-03-12
&Placement.Zone=ap-guangzhou-2
&ImageId=img-pmqg1cw7
&UserData=IyEvYmluL2Jhc2gKZWNoYAiSGVsbG8gVGluY2VudCBDbG91ZC4iCg==
&<Common Request Parameters>
```

Viewing execution logs

After the CVM instance is created successfully, you can run the following command to view the script execution log:

```
cat /var/log/cloud-init-output.log
```


Setting Custom Data (Windows CVM)

Last updated : 2024-12-24 17:37:14

Overview

When creating a CVM, you can configure an instance by specifying **custom data**. During the **first launch** of the CVM, the custom data will be passed into the CVM in text format and be executed. If you purchase multiple CVMs at a time, the custom data text will be executed on all CVMs during their first launch.

This document describes an example in which a PowerShell script is passed during the first launch of a Windows CVM.

Notes

For Windows Server 2012 R2 and later versions, custom data is supported.

A command can be executed by passing text only when a CVM is launched for the first time.

Before Base64 encoding, the size of the custom data cannot exceed 16 KB.

Custom data is Base64 encoded and then passed. If you directly copy a non-Base64 script file, do not select "The entry is Base64-encoded text".

During launch, executing specified tasks in custom data will increase the amount of time it takes to launch the CVM. We recommend that you wait for a few minutes, and after the tasks are completed, test whether the tasks have been successfully executed.

In this example, specify the Windows PowerShell script by using the PowerShell label, for example, the <powershell> </powershell> label.

Directions

Preparing text

Prepare text based on your actual requirements:

PowerShell script

Use the PowerShell label to prepare a PowerShell script.

For example, if you need to create a "tencentcloud.txt" file with the content of "Hello Tencent Cloud." in the C drive (C:), use the PowerShell label to prepare the following content:

```
<powershell>
```

```
"Hello Tencent Cloud." | Out-File C:\\tencentcloud.txt
</powershell>
```

Base64 encoded script

1. Run the following command to create a PowerShell script named “script_text.ps1”.

```
vi script_text.ps1
```

2. Press **i** to switch to the editing mode, refer to the following content, write it into the file, and save the “script_text.ps1” script.

```
<powershell>
"Hello Tencent Cloud." | Out-File C:\\tencentcloud.txt
</powershell>
```

3. Run the following command to perform the Base64 encoding operation on the “script_text.ps1” script.

```
base64 script_text.ps1
```

The following information is returned:

```
PHBvd2Vyc2h1bGw+CiJIZWxsbyBUZW5jZW50IENsb3VkLiIgfcBPdXQtRmlsZSAgQzpcdGVuY2VudGN
sb3VkLnR4dAo8L3Bvd2Vyc2h1bGw+Cg==
```

Passing text

We provide multiple methods to launch an instance, and here we introduce two of them. Choose a method according to your requirements:

Console

API

1. Purchase an instance (see [Creating an Instance](#)), and click **Advanced settings** under **Other settings** in 2.

Configure network and host.

1 Select basic configurations

2 Configure network and host

3 Confirm configuration

Network and bandwidth

Network Available IPs in the subnet: 253

If the existing VPCs/subnets do not meet your requirements, [create a VPC](#) or [a subnet](#) in the console. You can also change the VPC and subnet later.

Public network IP ☒ Get a free public IP

Bandwidth billing mode

By traffic

Bandwidth package

Note: The traffic fee is settled on an hourly basis. When your account balance becomes negative, the service will be stopped in 2 hours.

Bandwidth

1Mbps 10Mbps 20Mbps 100Mbps

5

 Mbps

Security group

Security group

New security group

Existing security group

Select security group(s) (up to five)

Please make sure port 22 (Linux SSH login) and port 3389 (Windows remote login) are open in the selected security group. You can go to the CVM console to [Modify security group](#)

Other settings

Tag

Tag key

Tag value

删除

+ Add

Instance name

(Optional) Defaults to "unnamed" if it's left empty. Automatic batch na

Supports batch sequential naming or pattern string-based naming. Up to 128 characters. 128 more characters are allowed.

Login methods

Set password

SSH key pair

Reset password after creation

Login name

root

Password

Enter the login password

type

Confirm password

Enter the login password

type

Note: Keep your password safe. Reset the password in the CVM console if necessary. Note that if you choose "Custom Password", the instance cannot be saved as a launch template.

Termination protection ☐ Prevent instances from being accidentally terminated in the console or via API

Security services ☒ Enable for free

Install the Cloud Workload Protection agent and activate CWP Basic for free

Cloud Monitor ☒ Enable for free

FREE cloud monitoring, analysis, alarming, and server monitoring metrics (component installation required)

Scheduled termination ☐ Enable scheduled termination

Enable it to terminate the CVM instance at the specified time

Advanced settings (hostname, CVM role, placement group, custom data)

2. In **Advanced settings**, enter the text content you have prepared in the **Custom data** text box.

PowerShell script: Type in [PowerShell script](#).

Base64 encoded script: Select "The above input is encoded with base64", and then type in [Base64 encoded text](#).

Advanced settings (hostname, CVM role, placement group, custom data) [^](#)

Hostname Supports batch sequential naming or pattern string-based naming

2-60 characters, including uppercase and lowercase letters, numbers, hyphens "-" and dots ".", It supports the {R:number} format, but colons ":" and braces "[]" are not allowed. Hyphens "-" and dots "." cannot be used consecutively, and cannot be placed at the beginning of the hostname. A number-only password is not allowed

Project

CAM role [Create a CAM role](#)

Placement group ☐ Add the instance to a placement group

If the existing placement groups are not suitable, please [create a new one](#)

Custom data

(Optional) It's used for configuration while launching an instance. It supports the Shell format. The size of original data is up to 16 KB. Shell script should start with #!, following by a path pointed to the parser to read the script (usually /bin/bash).

☐ The above input is encoded with base64.

3. Follow the instruction to complete CVM creation.

When creating a CVM by using API, you can pass the text by assigning the value of the encoded result returned in [Base64 encoded script](#) to the UserData parameter of the RunInstances API.

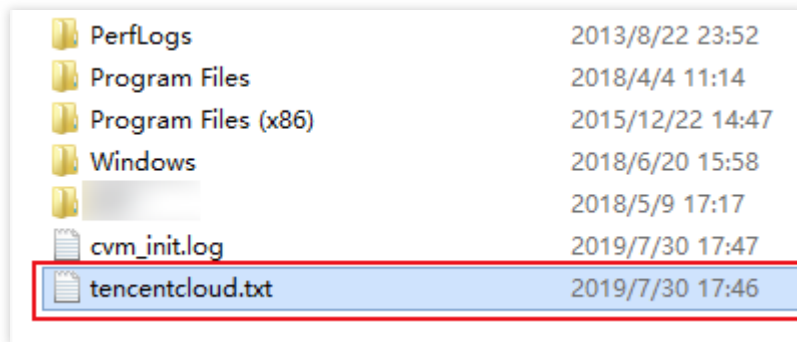
The following is an sample CVM creation request with UserData:

```
https://cvm.tencentcloudapi.com/?Action=RunInstances
&Version=2017-03-12
&Placement.Zone=ap-guangzhou-2
&ImageId=img-pmqg1cw7
&UserData=PHBvd2Vyc2h1bGw+CiJIZWxsbyBUZW5jZW50IENsb3VkLiIgZCBPdXQtRmlsZSAuXHRlbmN1b
&<Common Request Parameters>
```

Verifying custom data configuration

1. Log in to your CVM.
2. On the operating system interface, open the C drive (C:), and check whether the `tencentcloud.txt` text file exists.

If the `tencentcloud.txt` text file exists, the configuration is successful, as shown in the following figure:



PerfLogs	2013/8/22 23:52
Program Files	2018/4/4 11:14
Program Files (x86)	2015/12/22 14:47
Windows	2018/6/20 15:58
	2018/5/9 17:17
cvm_init.log	2019/7/30 17:47
tencentcloud.txt	2019/7/30 17:46

Viewing execution logs

You can view the `C:\Program Files\Cloudbase Solutions\Cloudbase-Init\log\cloudbase-init.log` file to get the execution logs of the script.

Work with Operating Systems

System Activation

Activating Windows with slmgr Command

Last updated : 2024-01-06 17:38:11

Overview

This document describes how to activate the operating system in a Windows CVM instance.

Note:

This document is intended only for Windows Server public images provided by Tencent Cloud. It is not applicable for custom images or images imported from external sources.

Directions

1. Log in to the Windows CVM instance. For more information, see [Logging in Using Standard Method \(Recommended\)](#).
2. Right-click in the lower left corner of the operating system's desktop,



, and select **Windows PowerShell (Admin)** from the pop-up menu.

3. In the PowerShell window, run the following commands in sequence to activate the operating system.

```
slmgr /upk
```

```
slmgr /ipk <ProductKey>
```

```
slmgr /skms kms.tencentyun.com
```

```
slmgr /ato
```

<ProductKey> in the `slmgr /ipk <ProductKey>` command should be replaced as follows:

Windows Server 2008 R2 Enterprise: 489J6-VHDMP-X63PK-3K798-CPX3Y

Windows Server 2012 R2 Datacenter: W3GGN-FT8W3-Y4M27-J84CP-Q3VJ9

Windows Server 2016: CB7KF-BWN84-R7R2Y-793K2-8XDDG

Windows Server 2019: WMDGN-G9PQG-XVVXX-R3X43-63DFG

Windows Server 2022 : WX4NM-KYWYW-QJJR4-XV3QB-6VM33

For more information, see [Key Management Services \(KMS\) client activation and product keys](#).

4. Restart the CVM instance to make the settings take effect. For more information, see [Restarting Instances](#).

FAQs

In some scenarios where the Windows operating system is not activated, the system memory of a high-end server will be limited to 2 GB, and the rest of the memory will be restricted in the form of "hardware reserved memory". The reason is that the `HKEY_LOCAL_MACHINE\\System\\CurrentControlSet\\Control\\ProductOptions` registry key is damaged. You can run the following command to determine whether to reactivate the system.

```
(Get-ItemProperty  
HKLM:\\SYSTEM\\CurrentControlSet\\Control\\ProductOptions\\).ProductPolicy.coun  
t
```

If the returned result is above 10,000, such as 56184, there is no need to reactivate the system.

If the returned result is "Unactivated value: 1960", refer to the following methods to solve the problem.

Method 1

Method 2

Method 3

1. Run the following command to activate the operating system.

```
slmgr.vbs /ipk <ProductKey>
```

Note:

Replace `<ProductKey>` according to the actual operating system version as detailed in [ProductKey](#).

2. After the command is executed, run the `Get-ItemProperty...` command again for verification. The returned value is 56184.

3. Restart the CVM instance to make the settings take effect. For more information, see [Restarting Instances](#).

4. Run the following command to activate the operating system.

```
slmgr.vbs /ato
```

1. Run the following command to fix it.

```
slmgr.vbs /rilc
```

2. After the command is executed, run the `Get-ItemProperty...` command again for verification, and the returned value is still 1960.

3. Run the following command to activate the operating system and restarting CVM.


```
slmgr.vbs /ato
```

1. Uninstall any MSI program.
2. Run the `Get-ItemProperty...` command again for verification, and the returned value may change. However, after the system is restarted, the memory limit is still 2 GB.
3. Run the following command to activate the operating system and restarting CVM.

```
slmgr.vbs /ato
```

Windows Server System Activation

Last updated : 2024-01-06 17:38:11

CVMs use KMS to authorize Windows servers.

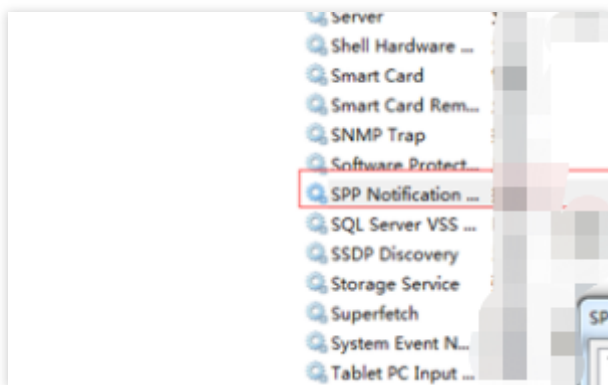
Note:

This document is intended only for Windows Server public images provided by Tencent Cloud. It is not applicable for custom images or images imported from external sources.

Windows Server 2008 and Windows Server 2012 need to be authorized using this method. The default KMS address (kms.tencentyun.com:1668) configured for the public images of Windows Server 2016 and Windows Server 2019 is correct and does not need to be modified.

Notes

1. Only the SPP Notification Service in **Windows Server 2008** is used for activation. Make sure it runs properly, as shown in the following figure:



2. Some optimization software may disable modifying the execute permissions of service programs. For example, modifying the execute permissions of sppsvc.exe might result in exceptions, as shown below:

sppnp.dll	2010/11/21 11:24	100 KB
sppobjs.dll	2010/11/21 11:24	1,058 KB
sppsvc.exe	2015/10/8 10:44	3,442 KB
sppuinotify.dll	2009/7/14 9:41	64 KB
sppwinob.dll	2010/11/21 11:24	409 KB
sppwmi.dll	2009/7/14 9:41	139 KB
sppreview.exe	2010/11/21 11:24	295 KB

Before you activate a Windows CVM, make sure that services and other features on the Windows CVM are normal.

Automatic Activation

Tencent Cloud encapsulates a script for activating Windows servers, which simplifies automatic activation. To use the activation script, follow the steps below:

1. Log in to the Windows CVM.
2. Download the [script](#), and run it to complete automatic activation.

Manual Activation

Notes

An inaccurate system clock will trigger an error during manual activation for some systems. In that case, you need to synchronize the system clock first by following the steps below:

Note:

If the system clock on the Windows CVM is normal, skip to [Activation](#).

1. Log in to the Windows CVM.
2. On the desktop, choose **Start > Run**. Enter `cmd.exe` in the **Run** dialog box to open the console window.
3. In the console window, run the following commands in sequence to synchronize the system clock:

```
w32tm /config /syncfromflags:manual /manualpeerlist:"ntpupdate.tencentyun.com"  
w32tm /resync
```

Activation directions

1. Log in to the Windows CVM.
2. On the desktop, choose **Start > Run**. Enter `cmd.exe` in the **Run** dialog box to open the console window.
3. In the console window, run the following commands in sequence to complete manual activation:

```
cscript /nologo %windir%/system32/slmgr.vbs -skms kms.tencentyun.com:1688  
cscript /nologo %windir%/system32/slmgr.vbs -ato
```

System Updates

Last updated : 2024-01-06 17:38:11

Introduction

This article uses Windows Server 2012 as an example to illustrate how to update your Windows operating system.

Directions

Updating Windows using the public network

You can use the Windows Update service to update your operating system. The detailed steps are as follows:

1. Log in to the Windows CVM instance.
2. Click



-> **Control Panel** -> **Windows Update**. The **Windows Update** window appears.

3. Click **Check for updates**, and wait until the check is finished.
4. After the check is finished, click **N Important Updates Available** or **N Optional Updates Available** in "Windows Update". The **Choose the Update to Install** window appears.
5. Select the updates you want to install, and click **Install**.

If you are prompted to restart your system after the update is completed, do so.

Note:

If you log in using VNC after the instance is rebooted and the message "Updating...Do not turn off the power" or "Configuration has not been completed" appears, do not perform a force-shutdown. That may damage your CVM instance.

Updating Windows using the private network

If your CVM instance does not have public network access, you can use the Tencent Cloud private Windows Update server to update your operating system. The Tencent Cloud Windows Update server has most of the updates, but does not have hardware driver packages and some uncommon updates. Therefore, uncommon services may not get updates using the Tencent Cloud Windows Update server.

Follow these steps to use the Tencent Cloud Windows Update server:

1. Log in to the Windows CVM instance.
2. Use Internet Explorer to download the Tencent Cloud private network configuration file wusin.bat from the following address:

<http://mirrors.tencentyun.com/install/windows/wusin.bat>

3. Run wusin.bat in Command Prompt with administrator privilege, as shown below:

Note:

If you open wusin.bat using Internet Explorer after it finishes downloading, the console window will automatically close and you cannot observe the output.

You can save wusin.bat to your hard drive, such as C:, and open a Command Prompt window with administrator privilege to run it.

If you no longer wish to use the Tencent Cloud Windows Update server, you can download wusout.bat clean it up.

Follow these steps to do so:

1. Log in to the Windows CVM instance.

2. Use Internet Explorer to download the Tencent Cloud private network configuration file wusout.bat from the following address:

<http://mirrors.tencentyun.com/install/windows/wusout.bat>

3. Run wusout.bat in Command Prompt with administrator privilege, as shown below:

Note:

If you open wusout.bat using Internet Explorer after it finishes downloading, the console window will automatically close and you cannot observe the output information.

You can save wusout.bat to your hard drive, such as C:, and open a Command Prompt window with administrator privilege to run it.

System Shutdown

Last updated : 2024-01-06 17:38:11

CVM Shutdown Process Analysis

Shutdown process

Note:

See [Shutdown Instances](#) for related operations.

The shutdown process of a Tencent Cloud Windows instance is as follows:

1. The host sends the shutdown command via libvirt on the QMP protocol to the `qemu` component.
2. The `qemu` component transfers the shutdown command to the CVM by interrupting ACPI (for more information, see technical documents on VMCS).
3. After receiving the shutdown signal, the Windows instance exits the applications and the service processes.
4. Close the core service process.
5. Turn off the power.

Note:

The sequence of closing the applications and the services in step 3-4 may vary by the system settings.

Windows is a closed-source system. It provides APIs that allow kernel-mode and user-mode programs to intervene in the shutdown process. Some running Windows services will also affect the shutdown process, increase the shutdown time, and cause shutdown failure.

Forced shutdown

In a virtualization scenario, in addition to shutdown methods such as the soft shutdown initialized by the system signal and message notification, you can also force the CVM to shut down via **forced shutdown**.

The forced shutdown may affect Windows and the user experience in the following two aspects:

1. A forced shutdown interrupts some services and applications and causes abnormal operations such as unsaved documents and unfinished WindowsUpdate processes.
2. During the shutdown process, the NTFS system (or the earlier FAT32 system) of Windows writes key data to the disk. A forced shutdown may result in write failure and cause Windows to believe the NTFS file system is damaged. Therefore, we recommend that you **first use soft shutdown** on a Windows instance.

Shutdown Failure Scenarios

The Windows system may have issues that affect the shutdown process and cause shutdown failure, including but not limited to:

1. A WindowsUpdate process may prolong the shutdown time. The Windows system may perform patch operations during the shutdown process and prompt a message like "Please do not power off or unplug your computer".
2. If the Windows system has the "Shutdown Event Tracker" mechanism enabled and needs to shutdown due to any system service or driver error, the system will require the user to submit an error description based on the configuration. The system will wait for you to complete these operations before shutdown.
3. Windows supports no shutdown without login. Under this configuration, the soft shutdown command sent from the virtual host will be discarded by Windows, so Windows will not shut down.
4. Before the shutdown, Windows will broadcast a message to every service and application. If the applications fail to give affirmative responses, Windows will not initiate the shutdown. In this case, you can configure Windows to ignore this response process.
5. If you configure the power management on Windows to ignore or do nothing **When I press the power button**, Windows will ignore the shutdown event received from the virtual host.
6. Based on the power management settings, Windows will go into the Sleep mode and ignore the shutdown event.
7. If the Windows system itself is damaged due to malicious software or infections with Trojans or other viruses, Windows may prevent shutdown.

Tencent Cloud has solved most of the shutdown failures when releasing the Windows public image to ensure soft shutdown. However, if your Windows is infected with viruses or Trojans, the system is damaged, or the Windows settings are adjusted again, the soft shutdown may fail.

Forced shutdown involves risks. Therefore, we recommend that you use it only when you really have to.

Configuring High-performance Power Management

Last updated : 2024-01-06 17:38:11

Overview

High-performance power management is required for the Windows Server operating system (OS) to support the soft shutdown of CVM instances. Otherwise, you can only forcibly shut down CVM instances on the CVM console. This document uses Windows Server 2012 as an example to describe how to configure power management.

Notes

To modify power management, you do not need to restart your computer.

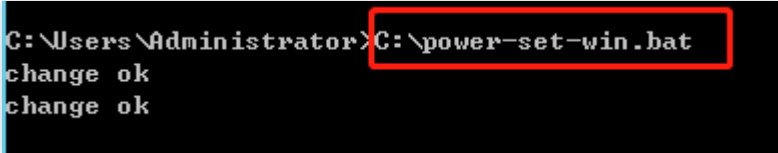
Directions

1. Log in to a Windows instance using either [the RDP file \(recommended\)](#) or [the remote desktop](#).
2. Open Internet Explorer in the Windows CVM, access the Tencent Cloud private network, and download the Tencent Cloud power modification and configuration tool.

The download address is `http://mirrors.tencentyun.com/install/windows/power-set-win.bat` .

For example, download the Tencent Cloud power modification and configuration tool (power-set-win.bat) to the C: drive.

3. Use the command line tool (CMD) as the administrator to open power-set-win.bat.



```
C:\Users\Administrator>C:\power-set-win.bat  
change ok  
change ok
```

4. Run the following command to view the current power management plan:

```
powercfg -L
```

The returned result is as follows:


```
C:\Users\Administrator>powercfg -L

Existing Power Schemes (* Active)
-----
Power Scheme GUID: 31b2f334-61e6-4a3c-b86f-334e6bb0431e (Balanced)
Power Scheme GUID: 54558641-32e2-436c-b370-1417f650a082 (High performance) *
Power Scheme GUID: 8c5eacf6-eabf-47a8-93d6-9b47f0e7687d (Power saver)

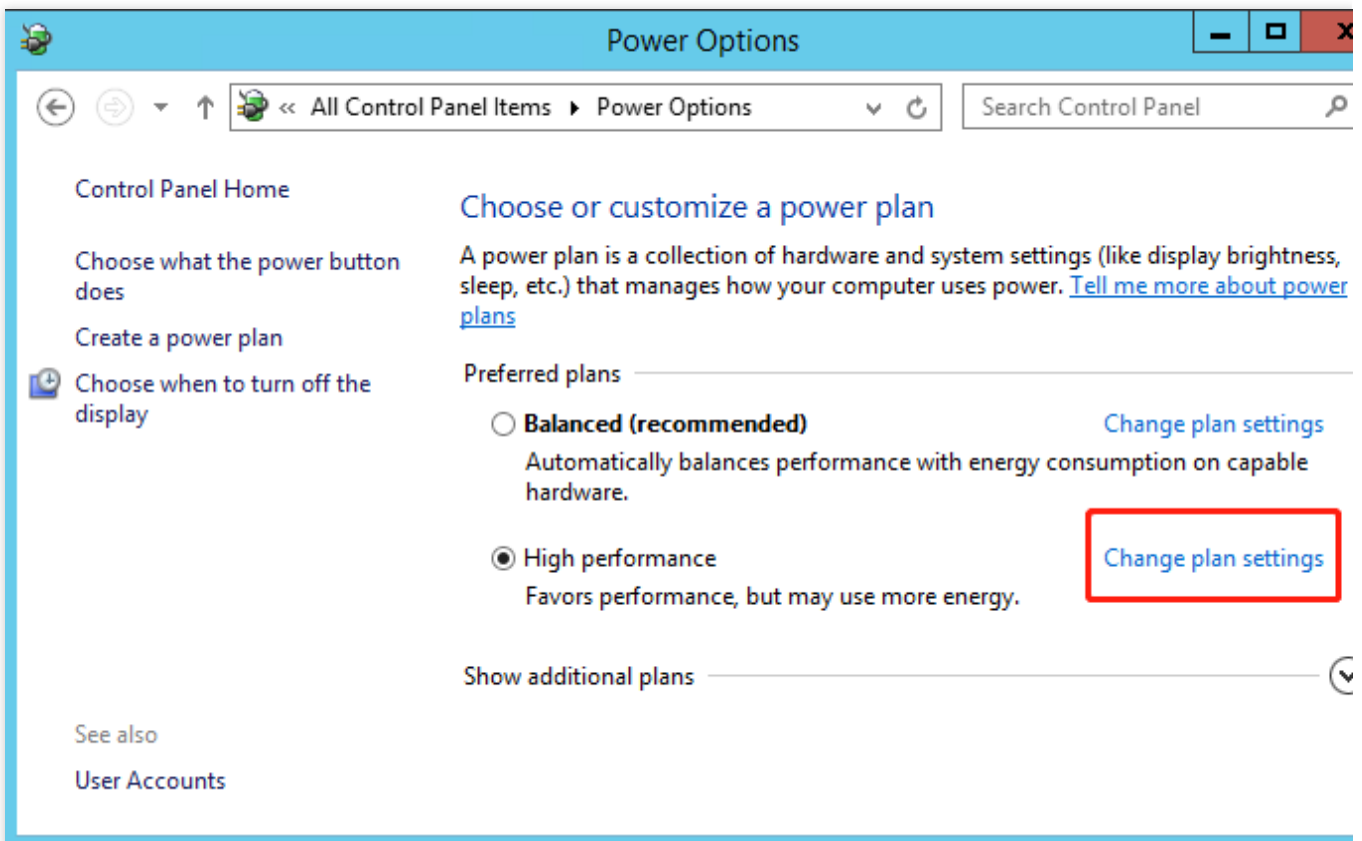
C:\Users\Administrator>
```

5. On the desktop of the operating system, click

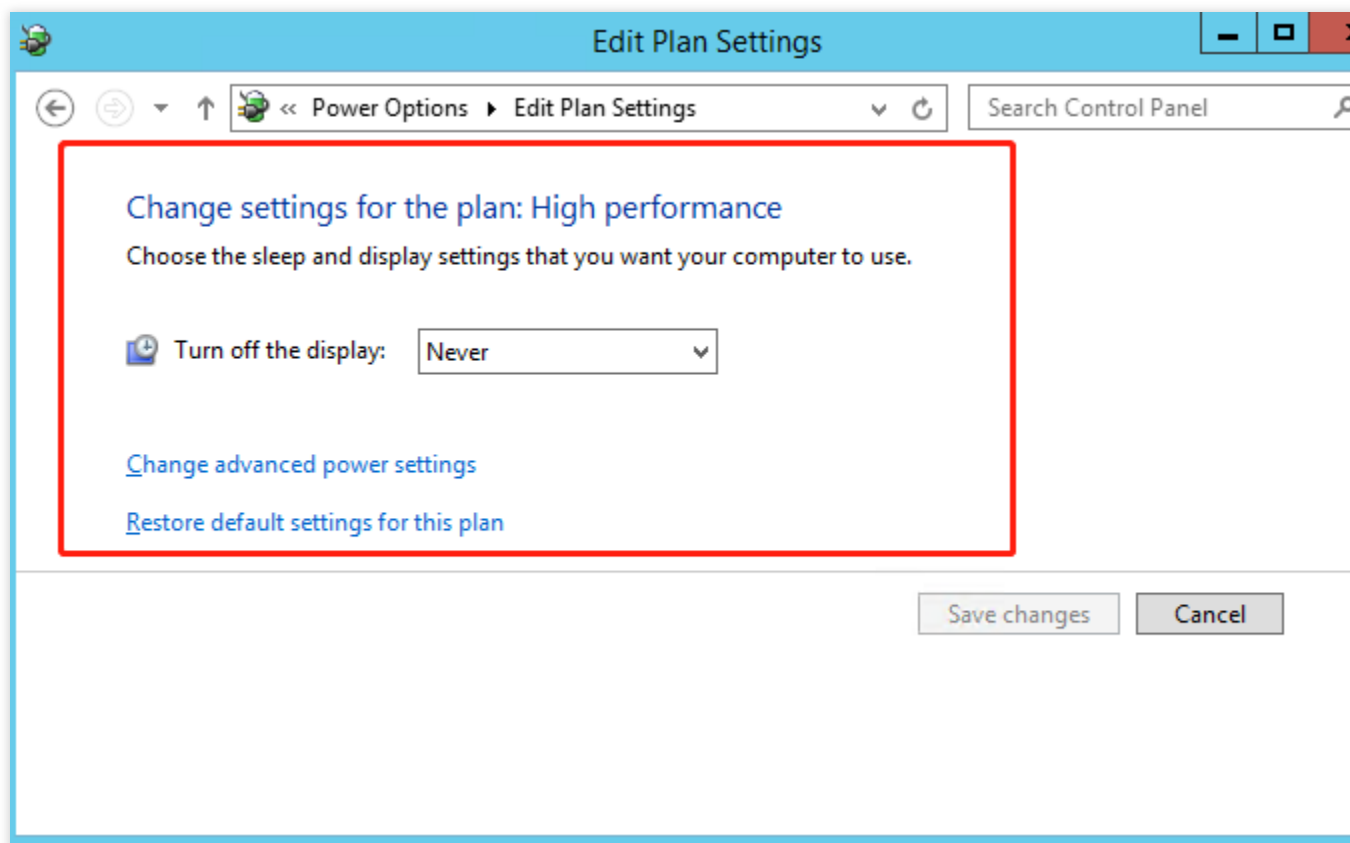


> **Control Panel** > **System and Security** > **Power Options**.

6. In the **Power Options** window, click **Change plan settings**.



7. In the **Edit Plan Settings** window, modify the idle turn-off times of the monitor and disk.



Windows Recovery Mode

Last updated : 2024-01-06 17:38:11

Windows Recovery Mode

Windows Recovery is an auto-recovery feature of Windows. When Windows detects certain system problems and believes that continuous running will cause system damage, this automatic repair feature will prevent Windows from starting up and provide **System Recovery Options** to users to repair, back up or restore the system.

System Recovery Options includes **Startup Repair**, **System Restore**, and **Windows Memory Diagnostic**, which can help users fix problems, back up data, and restore the system.

If you cannot remotely log in to a CVM, and see the following figure when logging in to the CVM via the console, it means that the Windows CVM has entered the recovery mode.



Reasons

In the following situations, the system may enter the recovery mode.

The power was cut off while Windows was running or shutting down. The Windows system was forcibly shut down via the console. Both scenarios may cause loss of critical data.

The power was cut off while Windows was updating. This may cause loss of critical update data.

The system was damaged by Trojans or viruses.

There were bugs in Windows core services. Windows detected a risk.

The system lost critical data or was damaged. The system files were corrupted due to misoperation.

Precautions

Recommended precautions are as follows:

Log in to the console to monitor the shutdown process of a Windows system. Tencent Cloud supports the soft shutdown that comes with a timeout mechanism. If the shutdown process has not been completed within the given period, a failure message will be returned. If the shutdown process is slow or Windows Update starts, do not force a shutdown; instead, just wait for the shutdown to complete. For more information, please see [Shutdown Failure Scenarios](#).

Check whether there are abnormal programs or processes in the system, such as Trojans or viruses.

Check whether the system management and anti-virus software are running properly.

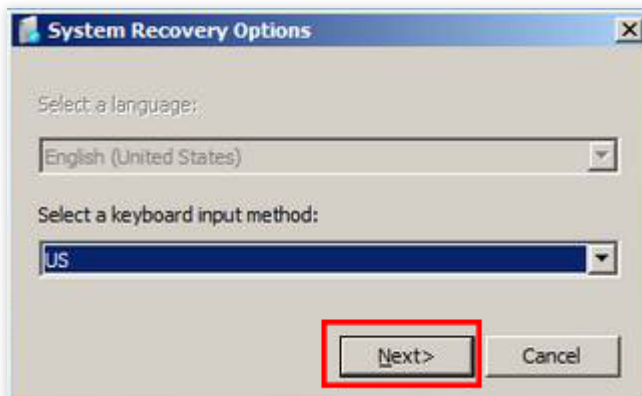
Install Windows updates in time, especially important updates and security updates.

Check the system event logs regularly to fix bug in core services.

Solutions

If Windows enters the recovery mode, perform the following steps to continue starting up or allow Windows to automatically repair minor problems.

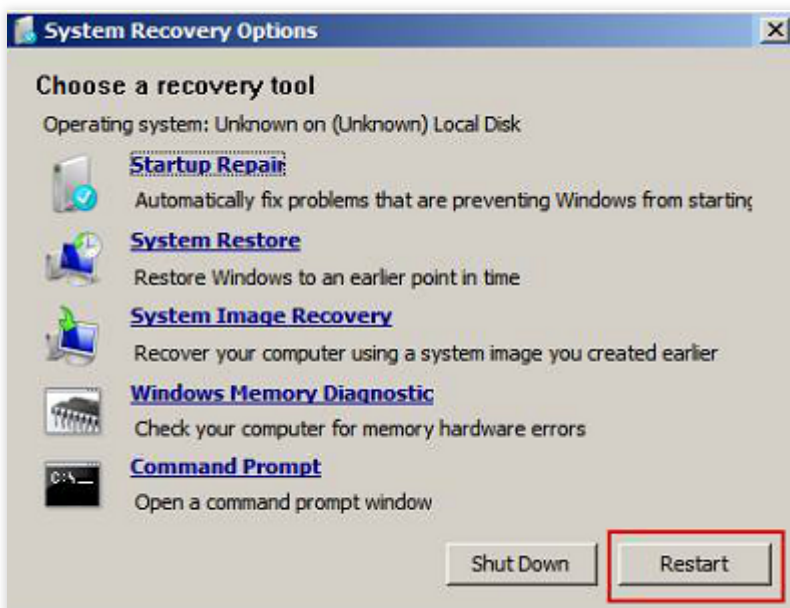
1. Log in to the CVM via the [CVM console](#).
2. In the recovery mode page that appears, click **Next**.



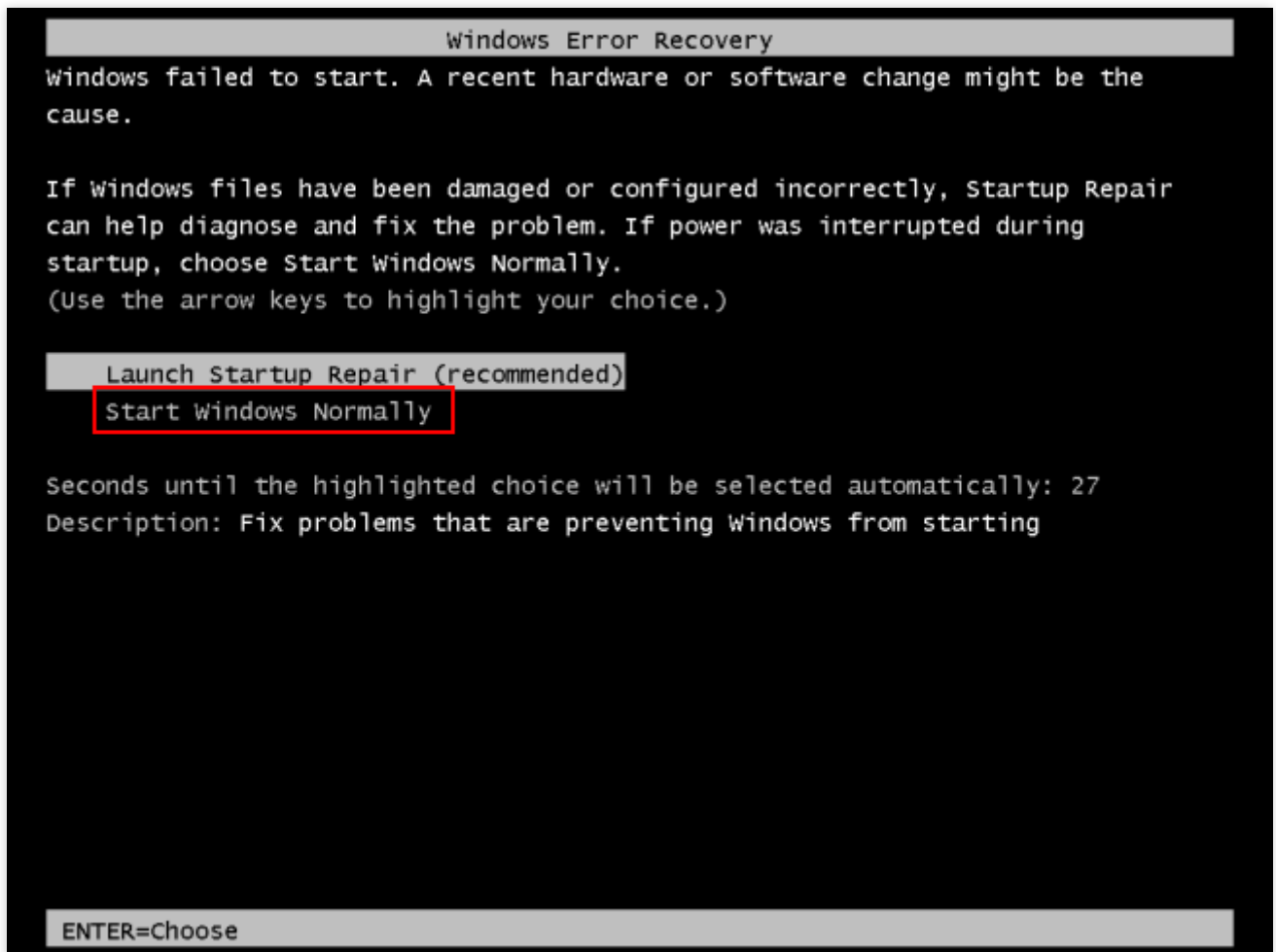
3. In the **System Recovery Options** window, click **Next** to use the default solution, as shown below:



4. Click **Restart**, and quickly press **F8**.



5. Choose **Start Windows Normally**.



If the startup fails, reinstall the system via the console. For more information, see [Reinstalling the system via the console](#).

Updating the Virtio ENI Driver

Last updated : 2024-01-06 17:38:11

Overview

For Tencent Cloud CVMs that using Windows Server 2008 R2, Windows Server 2012 R2, Windows Server 2016, and Windows Server 2019, Virtio ENI drivers are installed to optimize the network performance of virtualization hardware. This document describes how to update the Virtio ENI driver and check the driver version.

Prerequisites

You have logged in to a Windows CVM.

Directions

Viewing system version information

Perform the following steps to view the system version.

1. Log in to the CVM, right-click



, and select **Run** in the pop-up menu.

2. Enter `cmd` in the pop-up window and press **Enter**.

3. In the pop-up window, run the `systeminfo` command to view the system information.

This document uses Windows Server 2016 Datacenter Edition 64-bit as an example.

```
Administrator>systeminfo

OCCLOUD-VM
Microsoft Windows Server 2016 Datacenter
```

Updating the Virtio ENI driver

Caution:

The CVM will be disconnected briefly during the update. Please make sure that this will not affect your service before continuing. The CVM needs to be restarted after the update.

1. Download the Virtio ENI driver installation file for the operating system version by using the browser in the CVM.

Choose the download address based on the network environment:

Public network download address:

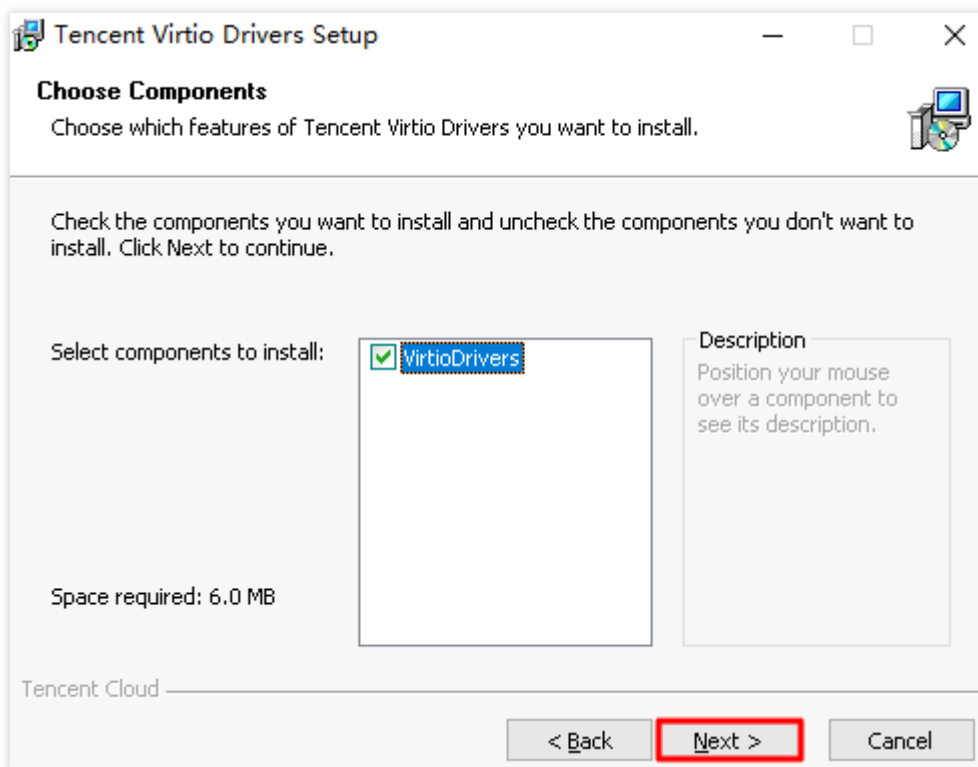
```
http://mirrors.tencent.com/install/windows/virtio_64_1.0.9.exe
```

Private network download address:

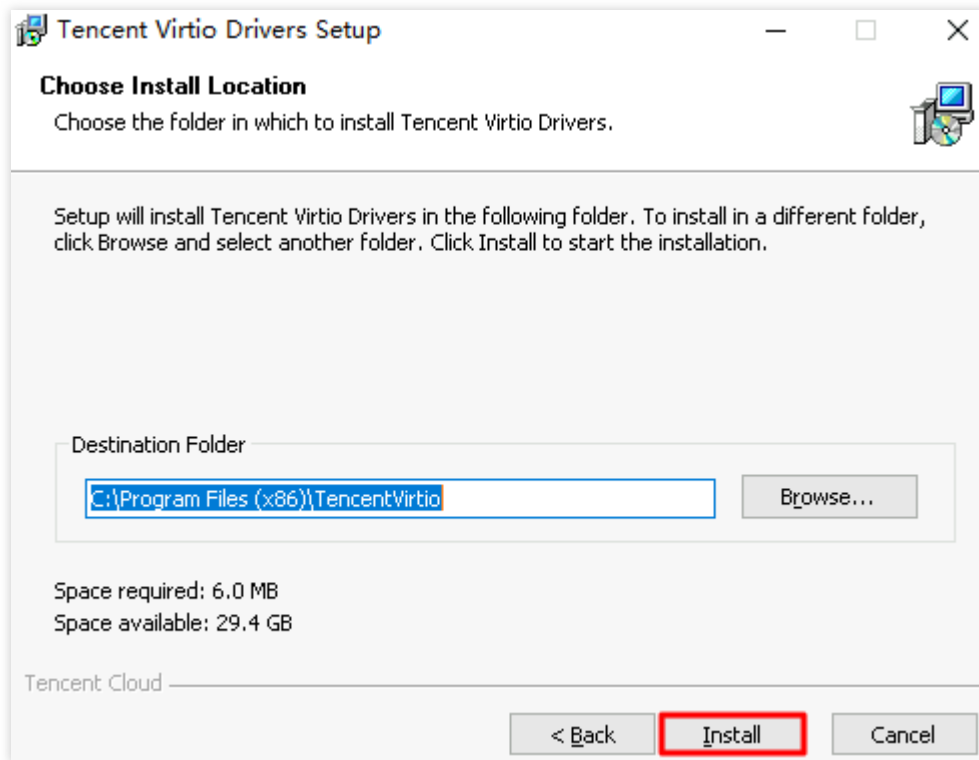
```
http://mirrors.tencentyun.com/install/windows/virtio_64_1.0.9.exe
```

2. After the download is completed, double-click the installer and click **Next**.

3. Keep "VirtioDrivers" checked by default and click **Next**, as shown below:



4. Select the installation location and click **Install**, as shown below:



5. In the Windows Security pop-up window, check **Always trust the software from Tencent Technology (Shenzhen) Company Limited** and click **Install**.

During the installation process, choose **Install this driver software anyway** when prompted.

6. Follow the prompt to restart the computer to complete the update.

Checking the driver version

1. Right-click



and select **Run** in the pop-up menu.

2. In the **Run** window, enter **ncpa.cpl** and press **Enter**.

3. In the **Network Connection** window, right-click the "Ethernet" icon and choose **Properties**.

4. In the **Ethernet Properties** window, click **Configure**.

5. In the **Tencent Virtio Ethernet Adapter Properties** window, select the **Driver** tab to see the current driver version.

Modifying SID

Last updated : 2024-01-06 17:38:11

Overview

Windows operating system uses a security identifier (SID) to identify computers and users. CVM instances created with the same image will have the same SID and might experience domain log-in issues. If you need to build a Windows domain environment, you will need to modify the SID.

This document takes the CVM with Windows Server 2012 operating system as an example to describe how to modify the SID using sysprep and sidchg tools of the system.

Must-knows

Instruction in this document is only applicable to Windows Server 2008 R2/2012/2016.

To modify the SIDs of multiple CVMs at the same time, you can create a custom image (select "run sysprep to create the image").

Modify the SID may cause data loss or system damage. We recommend you create a system disk snapshot or image in advance.

Directions

Using sysprep to modify the SID

Note:

After using sysprep to modify the SID, you need to manually reset some system parameters, including IP configuration information.

When you use sysprep to modify the SID, C:\Users\Administrator will be reset and some data on the system disk will be erased. Please back up data.

1. [Log in to the Linux CVM via VNC](#).
2. In the operating system interface, right-click



> **Run**, enter **cmd**, and press **Enter** to open the admin command line.

3.

In the admin command line, run the following command to save the current network configuration.

```
ipconfig /all
```

4. In the admin command line, run the following command to open the sysprep tool.

```
C:\\Windows\\System32\\Sysprep\\sysprep.exe
```

5. In the **System Preparation Tool 3.14** window that pops up, make the following configurations.

Set **System Cleanup Action** to **Enter System Out-of-Box Experience (OOBE)** and check **General**.

Set **Shutdown Options** to **Reboot**.

6. Click **OK**, and the system restarts automatically.

7. After the startup is complete, follow the wizard to complete the configuration (select language, reset password, etc.).

8. In the operating system interface, right-click



> **Run**, enter **cmd**, and press **Enter** to open the admin command line.

9. Run the following command to verify whether the SID has been modified.

```
whoami /user
```

If a message similar to the following is returned, the SID is modified.

```
C:\Users\Administrator>whoami /user
[redacted]
-----
[redacted]
=====
win-...\\administrator S-1-5-21-200470050-3688556951-3253297147-500
```

10. Reset ENI information (such as IP address, gateway address, DNS, etc.) based on the network configuration information saved in [Step 3](#).

Using sidchg to modify the SID

1. Log in to your CVM.

2. Access and download the sidchg tool through IE browser.

The download address of sidchg tool: <http://www.stratesave.com/html/sidchg.html>

3. Using the admin command line to run the following command and open the sidchg tool.

In this example, the sidchg tool is saved on C: drive and named as "sidchg64-2.0p.exe".

```
C:\Users\Administrator>C:\sidchg64-2.0p.exe /R
SID Change Utility SIDCHG64 2.0p -- Copyright Stratesave Systems 2019
Shareware - visit http://www.stratesave.com for Licence and pricing

Enter license key or trial key:
=
```

`/R` means automatic restart after modification, and `/S` means shutdown after modification. For details, please see [SIDCHG Official Instructions](#).

4. Enter license key or trial key as prompted by the page, and press **Enter**.

5. Enter **Y** as prompted by the page, and press **Enter**.

```
C:\Users\Administrator>C:\sidchg64-2.0p.exe /R
SID Change Utility SIDCHG64 2.0p -- Copyright Stratesave Systems 2019
Shareware - visit http://www.stratesave.com for Licence and pricing

Enter license key or trial key:
34D6g-4sEJk-voWtg-4v
Temporary trial-key for evaluation only
To assure correct change of SID, current user will be logged out and SID change
will be done in background,
after which the system will reboot
Do not turn off or shut down your computer and do not Log into your computer whi
le SID change is running!!

Changing SID risks data loss and damaged System. Do you want to continue <Y/N>?
```

6. In the prompt box for modifying the SID, click **OK** to reset the SID.

The system restarts during the reset.

SID of this computer is being changed after
which the computer will reboot
Do not log on to this computer and do not turn
off the computer at this time !

7. After the startup is completed, right-click



> **Run**, enter **cmd**, and press **Enter** to open the admin command line.

8. Run the following command to verify whether the SID is modified.

```
whoami /user
```

If a message similar to the following is returned, the SID is modified.

```
C:\Users\Administrator>whoami /user  
[redacted]  
-----  
[redacted]  
===== SID =====  
win-[redacted]\administrator S-1-5-21-200470050-3688556951-3253297147-500
```

Modifying VNC Resolution

Last updated : 2024-01-06 17:38:11

Overview

This document introduces how to adjust the resolution of the instance login via VNC through the CVM console. If Windows system images have very low VNC resolutions, some applications may fail to be properly displayed or opened. To avoid these issues, you need to modify the resolutions of Windows system images.

Some Linux system images have lower VNC display resolutions by default. For example, the VNC resolution for CentOS is only 720 * 400. You can set the VNC resolution to 1024 * 768 by modifying the parameter `grub`.

Note:

More recent Linux images such as CentOS 7, CentOS 8, Ubuntu, and Debian 9.0 adopt a default VNC resolution of 1024 * 768, which can be used without any modifications.

Prerequisites

You have logged in to the instance via VNC. For details, see the following documents:

[Logging into Windows Instances via VNC.](#)

[Logging into Linux Instances via VNC.](#)

Directions

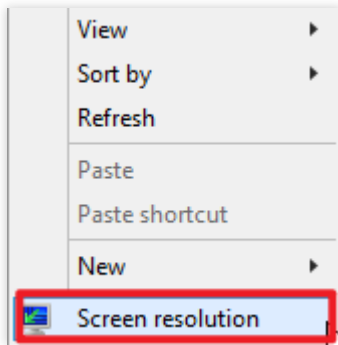
Windows instances

Linux instances

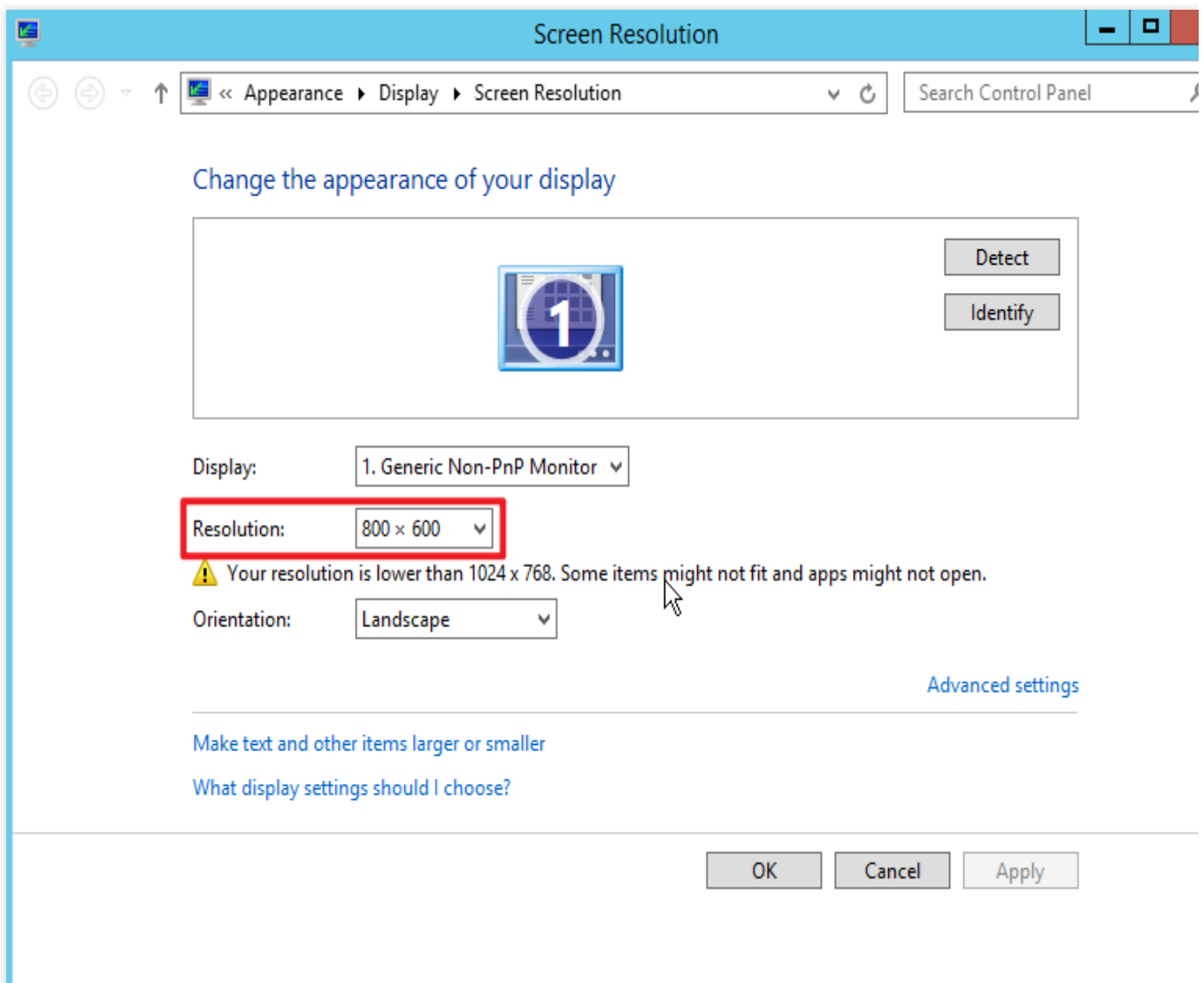
Note:

The steps below describe how to modify the VNC resolution of a Windows instance on the Windows Server 2012 operating system.

1. Right-click on the desktop and select **Screen resolution**, as shown below:



2. In the “Screen Resolution” window, set **Resolution** to your desired value and click **Apply**, as shown below:



3. In the pop-up window, click **Keep changes**.

4. Click **OK** to close the “Screen Resolution” window.

The steps below describe how to modify the VNC resolution of a Linux instance on the CentOS 6 and Debian 7.8 operating systems.

CentOS 6

A CentOS 6 image has a VNC resolution of 720 * 400 by default. To set its resolution to 1024 * 768, modify the launch parameter `grub` as follows:

1. Run the following command to open the `/etc/grub.conf` file.

```
vi /etc/grub.conf
```

2. Press `i` to switch to the editing mode, and add `vga=792` to the parameter `grub`, as shown below:

```
default=0
timeout=5
splashimage=(hd0,0)/boot/grub/splash.xpm.gz
hiddenmenu
serial --unit=0 --speed=9600 --word=8 --parity=no --stop=1
terminal --timeout=3 console serial
title CentOS (2.6.32-642.6.2.el6.x86_64)
  root (hd0,0)
  kernel /boot/vmlinuz-2.6.32-642.6.2.el6.x86_64 ro vga=792 root=/dev/vda1 console=ttyS0 console=tty0 panic=5 rd_NO_LUKS rd_NO_LVM LANG=C rd_NO_MD SYSFONT=latarcyrheb-sun16 crashkernel=auto KEYBOARDTYPE=pc KEYTABLE=us rd_NO_DM
  initrd /boot/initramfs-2.6.32-642.6.2.el6.x86_64.img
title CentOS 6 (2.6.32-642.el6.x86_64)
  root (hd0,0)
  kernel /boot/vmlinuz-2.6.32-642.el6.x86_64 ro vga=792 root=/dev/vda1 console=ttyS0 console=tty0 panic=5 rd_NO_LUKS rd_NO_LVM LANG=en_US.UTF-8 rd_NO_MD SYSFONT=latarcyrheb-sun16 crashkernel=auto KEYBOARDTYPE=pc KEYTABLE=us rd_NO_DM
  initrd /boot/initramfs-2.6.32-642.el6.x86_64.img
```

3. Click **Esc** and enter `:wq` to save and close the file.
4. Run the following command to restart the CVM.

```
reboot
```

Debian 7.8

Both Debian 7.8 and Debian 8.2 images have VNC resolutions of 720 * 400 by default. To set their resolutions to 1024 * 768, modify the launch parameter `grub` as follows:

1. Run the following command to open the `grub` file.

```
vim /etc/default/grub
```

2. Press `i` to switch to the editing mode and add `vga=792` to the end of the parameter value `GRUB_CMDLINE_LINUX_DEFAULT`, as shown below:

```
GRUB_DEFAULT=0
GRUB_TIMEOUT=5
GRUB_DISTRIBUTOR=`lsb_release -i -s 2> /dev/null || echo Debian`
GRUB_CMDLINE_LINUX_DEFAULT="crashkernel=1800M-4G:128M,4G-:168M panic=5 vga=792"
GRUB_CMDLINE_LINUX="console=ttyS0,9600n8 console=tty0"
GRUB_SERIAL_COMMAND="serial --speed=9600 --unit=0 --word=8 --parity=no --stop=1"
```

3. Click **Esc** and enter `:wq` to save and close the file.
4. Run the following command to update the `grub.cfg` file.


```
grub-mkconfig -o /boot/grub/grub.cfg
```

5. Run the following command to reboot the CVM.

```
reboot
```

Appendix

The VGA parameter values corresponding to different Linux instance resolutions are listed below:

Resolution	640 * 480	800 * 600	1024 * 768
VGA	786	789	792

Ensuring Unique SIDs for CVMs Using Sysprep

Last updated : 2024-01-06 17:38:11

Overview

To join an instance to a domain and log in to Windows CVM with the domain account, you need to run Sysprep before creating a custom image to ensure the SID will be unique. Otherwise, the instance cannot be joined to the domain because the CVM created with the custom image and the original instance contains identical information, such as duplicate SIDs. Skip this operation if you have no need to join your Windows CVM to a domain.

This document describes how to run Sysprep on the Windows Server 2012 R2 64-bit operating system to ensure that the Windows CVMs in the domain have unique SIDs.

For more information about Sysprep, see [https://technet.microsoft.com/zh-cn/library/cc721940\(v=ws.10\).aspx](https://technet.microsoft.com/zh-cn/library/cc721940(v=ws.10).aspx).

Notes

Windows CVM must be an active genuine Windows operating system.

If your Windows CVM is created with a non-public image, you must always run the Sysprep version provided in the original image under directory `%WINDIR%\system32\sysprep`.

The remaining Windows reset counts must be greater than 1, otherwise Sysprep cannot be encapsulated.

To check the remaining Windows reset counts, execute the command `slmgr.vbs /dlv`.

The Cloudbase-Init account takes care of the initialization actions when a CVM instance is launched. If you change/delete this account or uninstall Cloudbase-Init, you might lose custom information when you provision a new CVM from a custom image. Thus, we do not recommend modifying or deleting the Cloudbase-init account.

Prerequisites

[Log in to Windows CVM](#) as Administrator.

[Install Cloudbase-Init on Windows CVM](#).

Directions

1. On the desktop, click



to open a Windows PowerShell window.

2. Execute the following command in the Windows PowerShell window to go to the installation directory of the Cloudbase-init tool.

Note:

Assume that Cloudbase-init is installed under `C:\\Program Files\\Cloudbase Solutions\\`.

```
cd 'C:\\Program Files\\Cloudbase Solutions\\Cloudbase-Init\\conf'
```

3. Execute the following command to encapsulate the Windows system.

Note:

Include `/unattend:Unattend.xml` in the following command, otherwise your current username, password and other configurations of the CVM will be reset. If you choose **Follow image** for the login, you will need to manually reset the username and account after launch.

After the following command is executed, CVM automatically shuts down. To ensure that the CVMs created from this image has unique SIDs, do not launch the CVM instance before creating a custom image, or this action will only be effective for the current CVM.

After you execute the following command on the Windows Server 2012 or Windows Server 2012 R2 operating system, the account Administrator and its password of the CVM will be deleted. Reset your account and password after restarting the CVM as detailed in [Reset Instance Password](#).

```
C:\\Windows\\System32\\sysprep\\sysprep.exe /generalize /oobe  
/unattend:Unattend.xml
```

4. Create a custom image from the CVM instance on which Sysprep has been executed, and start new CVM instances with the image. See [Creating Custom Images](#).

And now, each new CVM instance will have a unique SID when joining a domain.

Note:

To check CVM SID, execute the command `whoami /user`.

Use of the atop Monitoring Tools by Linux Instances

Last updated : 2024-01-06 17:38:11

Overview

Atop is a tool for monitoring Linux system resources and processes. It records the system operation status at a certain frequency, collects the data of system resources (CPU, memory, disk and network) usage and process running status, and stores these information in the disk in the format of log files. You can obtain the corresponding atop log file to analyze an instance issue.

This document introduces how to use the atop monitoring tool with an example of CVM using CentOS 7.9 operating system.

Directions

Installing Atop

1. [Log in to a Linux instance using WebShell](#).
2. Run the following command to install atop.

```
yum install atop -y
```

The prompt message `Complete!` indicates that atop has been successfully installed.

Configuring and launching atop

Configure the atop monitoring period and log retention time as described below:

1. Execute the following command to open the atop configuration file with VIM editor.

```
vim /etc/sysconfig/docker
```

2. Press `i` to enter the edit mode and modify the configuration below:

Modify `LOGINTERVAL=600` to `LOGINTERVAL=30` . This means to modify the monitoring period of 600s by default to a recommended value-30s. You can modify it to other values as needed.

Modify `LOGGENERATIONS=28` to `LOGGENERATIONS=7` . This means to modify the log retention time of 28 days by default to 7 days. To prevent overmuch disk storage space from being occupied due to the long-term running of atop, it's recommended to modify it to 7 days. You can modify it to other values as needed.

The result should be as follows:

```
LOGOPTS=""  
LOGINTERVAL=30  
LOGGENERATIONS=7  
LOGPATH=/var/log/atop
```

3. Press **Esc** and enter **:wq** to save the modification and exit VIM editor.
4. Launch atop with the following commands executed in CentOS 7 and other upgraded systems.

```
systemctl restart atop
```

Analyzing atop

The launched atop records the collected data in the log file under the `/var/log/atop` directory. Please check the required log file name, execute the following command to view the log file, and analyze the data with reference to [atop Common Commands](#) and [System Resource Monitoring Field Description](#).

```
atop -r /var/log/atop/atop_2021xxxx
```

atop Common Commands

You can use the following commands to filter the required data after opening the log file:

- c**: Filter by CPU utilization of Linux processes in descending order.
- m**: Filter by memory usage of Linux processes in descending order.
- d**: Filter by disk usage of Linux processes in descending order.
- a**: Filter by resource usage of Linux processes in descending order
- n**: Filter by network usage of Linux processes in descending order (This command is not supported by default. To use it, you need to install additional kernel modules).
- t**: Redirect to the next monitoring data collection spot.
- T**: Redirect to the previous monitoring data collection spot.
- b**: Specify point in time, in the format YYYYMMDDhhmm.

Description of system resource monitoring fields

Some monitoring fields and values are shown below. These values obtained according to the sampling period are only for reference.

ATOP - VM-55-10-centos					2021/09/08 17:33:41					-----					20d5h22m16s el				
PRC	sys	55m55s	user	58m56s	#proc	89	#trun	1	#tslpi	117	#tslpu	0	#zombie	0	clones	423e4	#exit		
CPU	sys	1%	user	1%	irq	0%	idle	98%	wait	0%	steal	0%	guest	0%	curf	2.49GHz	curscal		
CPL	avg1	0.09	avg5	0.03	avg15	0.01					csw	154235e4	intr	29732e4			numcpu		
MEM	tot	1.8G	free	264.8M	cache	739.9M	dirty	0.5M	buff	656.5M	slab	97.8M							
SWP	tot	0.0M	free	0.0M												vmcom	882.0M	vmlim	938
PAG	scan	0	steal	0	stall	419										swin	0	swout	
DSK		vda	busy	0%	read	17941	write	2962e3	KiB/r	12	KiB/w	9	MBr/s	0.00	MBw/s	0.02	avio	0.99	
NET	transport		tcpi	6746364	tcpo	6489494	udpi	321203	udpo	316716	tcpao	845510	tcpo	28844	tcprs	13139	udpip		
NET	network		ipi	8334300	ipo	8060255	ipfrw	0	deliv	8334e3					icmpi	1227e3	icmpo	124	
NET	eth0	----	pcki	8653672	pcko	8379631	si	4 Kbps	so	5 Kbps	erri	0	erro	0	drpi	0	drpo		
*** system and process activity since boot ***																			
PID	TID	RUID	EUID	THR	SYSCPU	USRCPU	VGROW	RGROW	RDDSK	WRDSK	ST	EXC	S	CPUNR	CPU	CMD			
2094	-	root	root	4	12m50s	24m58s	655.9M	14228K	20K	650.9M	N-	-	S	0	0%	barad_age			
5339	-	root	root	5	8m53s	13m08s	20848K	6692K	0K	57304K	N-	-	S	0	0%	tat_agent			
8937	-	root	root	9	13m40s	4m07s	1.0G	17304K	0K	38340K	N-	-	S	0	0%	YDLive			
1	-	root	root	1	9m02s	8m40s	19364K	1632K	343.2M	4.3G	N-	-	S	0	0%	init			
2093	-	root	root	1	60.31s	5m06s	153.7M	9632K	4K	243.8M	N-	-	S	0	0%	barad_age			
4140	-	root	root	26	2m57s	1m50s	1.0G	37228K	24K	95296K	N-	-	T	0	0%	YDService			
24	-	root	root	1	2m15s	0.00s	0K	0K	0K	0K	N-	-	S	0	0%	ata_sff/0			
4	-	root	root	1	2m00s	0.00s	0K	0K	0K	0K	N-	-	S	0	0%	ksoftirqd			
4220	-	root	root	11	29.90s	45.23s	636.3M	14852K	0K	71688K	N-	-	S	0	0%	YDEdr			
7	-	root	root	1	67.29s	0.00s	0K	0K	0K	0K	N-	-	S	0	0%	events/0			
575	-	root	root	1	33.42s	0.00s	0K	0K	132K	3.5G	N-	-	S	0	0%	kjournald			
1490	-	root	root	2	6.37s	4.92s	29764K	880K	12K	262.2M	N-	-	S	0	0%	auditd			
7976	-	root	root	2	7.37s	3.35s	98724K	1940K	32K	138.2M	N-	-	S	0	0%	sgagent			
1512	-	root	root	4	3.68s	5.88s	245.7M	5484K	396K	186.5M	N-	-	S	0	0%	rsyslogd			
561	-	root	root	1	9.38s	0.00s	0K	0K	0K	0K	N-	-	S	0	0%	flush-252			
7836	-	root	root	1	8.05s	1.00s	66288K	1200K	316K	57452K	N-	-	S	0	0%	sshd			
8065	-	root	root	1	6.80s	1.71s	114.2M	1320K	84K	0K	N-	-	S	0	0%	crond			

The main parameters are as follows:

ATOP: CVM name, information sampling date and point in time.

PRC: Overall operation status of the process.

sys and user: The ratios between CPU service time and the sampling period in Linux process kernel mode and user mode.

#proc: Total Linux processes.

#zombie: Zombie processes.

#exit: Processes exited during the atop sampling period.

CPU line: Usage of overall CPU configuration (Deem multi-core CPUs as a complete CPU resource). All values of the fields on the CPU line are added up to N00%, and N is equal to the number of CPU cores.

sys and user: The ratios between CPU service time and the sampling period in process kernel mode and user mode.

irq: The ratio between the time the CPU is used to process interrupts and the sampling period.

idle: The ratio between the CPU idle time and the sampling period.

wait: The ratio between the time the CPU is in the "idle state caused by the process waiting for the disk Input/Output" and the sampling period.

CPL line: CPU loading conditions.

avg1、avg5 and avg15 : Average number of processes in the running queue in the past 1 minute, 5 minutes and 15 minutes.

csw: Number of context switches.

intr: Number of interruptions.

MEM line: Memory usage.

tot: Total physical memory.

cache: Memory used for page cache.

buff: Memory used for file buffer.

slab: Memory used by system kernel.

SWP line: Usage of swap space.

tot: Total swap area.

free: Idle swap space.

PAG: Paging of virtual memory

swin and swout: Number of memory pages swapped in and out.

DSK: Disk usage. A line of information shows the usage of each disk device. Add a line of DSK information if there is sdb device.

sda: Disk device identifier.

busy: Disk busy time percentage.

read and write: Number of reading and writing requests.

NET line: The multiple NET lines show the network conditions, including the information of transport layer (TCP and UDP), IP layer and active network interfaces.

xxxxxi: Number of packages received at each layer or active network interface.

xxxxxo: Number of packages sent at each layer or active network interface.

Stop atop

It is not recommended to run atop for a long time in application environment. You can stop atop after troubleshooting. In CentOS 7 and other upgraded systems, you can execute the following command to stop the atop.

```
systemctl stop atop
```

Introduction to Linux Kernel Parameters

Last updated : 2024-01-06 17:38:11

Tencent Cloud provides the Linux public images with default configurations, but we recommend you separately configure `sysctl` to adapt to your specific business. This document describes the default and optimal configurations of Tencent Cloud Linux public images and helps you manually tune them as needed.

Note:

The parameters expressed as “-” under **Initial Configuration** use the official image’s default configurations. The `sysctl -w` command only makes the configurations take effect temporarily, while parameters wrote to `/etc/sysctl.conf` take effect permanently.

Network-related

Parameter	Description	Initial Configuration
<code>net.ipv4.tcp_tw_recycle</code>	This parameter is used to quickly recycle the TIME_WAIT connection. If enabled, kernel will check the packet timestamp. We do not recommend enabling this parameter, because packet loss may occur when the timestamp is not monotonically increasing. This parameter is disused in later kernel versions.	0
<code>net.core.somaxconn</code>	This parameter is used to define the ESTABLISH state at the end of the three-way handshake when there is no ACCEPT queue. A longer	128

	<p>ACCEPT queue indicates low processing rate of the client, or a burst of new connections in a short time.</p> <p>Setting <code>net.core.somaxconn</code> too low may cause packet loss because the new SYN connection will be discarded when the server receives SYN packet and the <code>somaxconn</code> table is full. Setting it too high is only necessary for high concurrence service, but the latency may increase.</p>	
<code>net.ipv4.tcp_max_syn_backlog</code>	<p>This parameter specifies the maximal number of connections in SYN_RECV queue, which was once used to defend the common synflood attacks. However, if <code>tcp_syncookies=1</code>, connections in SYN_RECV queue will exceed the upper limit.</p>	-
<code>net.ipv4.tcp_syncookies</code>	<p>This parameter is used to enable SYN Cookies, which prevents some SYN attacks. If enabled, the connections can still be established</p>	1

	<p>when the SYN queue is overflowed. However, SHA1 will be used to verify Cookies, which theoretically increases the CPU utilization.</p>	
<pre>net.core.rmem_default net.core.rmem_max net.ipv4.tcp_memnet.ipv4.tcp_rmem</pre>	<p>These parameters are used to configure the cache size of received data. Setting this too high may waste memory resources, while setting it too low may cause packet loss. You can tune them according to the concurrence and throughput of your business.</p> <p>rmem_default: the theoretically optimal configuration is equal to the value of bandwidth divided by RTT, which will overwrite the configurations of tcp_rmem and tcp_rmem</p> <p>rmem_max: approximately five times of rmem_default</p> <p>tcp_mem: total TCP memory consumed, which is automatically set to to 3/32, 1/8 or 3/16 of the CVM's</p>	<p>rmem_default=655360rme</p>

	available memory. The parameters <code>tcp_mem</code> and <code>rmem_default</code> also determine the maximum number of concurrent connections.	
<pre>net.core.wmem_default net.core.wmem_max net.ipv4.tcp_wmem</pre>	These parameters are used to configure the data transmission cache. Data sending on Tencent Cloud usually does not have bottlenecks, so these configurations are optional.	-
<pre>net.ipv4.tcp_keepalive_intvl net.ipv4.tcp_keepalive_probes net.ipv4.tcp_keepalive_time</pre>	These parameters are relevant to TCP Keepalive, which default to 75/9/7200. The default settings mean that the kernel will initiate detection when a TCP connection is idle for 7,200 seconds, and will send RST after 9 failed detections (each for 75 seconds). These values are too high for a server. You can adjust them to 30/3/1800 as needed.	-
<pre>net.ipv4.ip_local_port_range</pre>	This parameter is used to configure the available port	-

	range, which can be adjusted as needed.	
<code>tcp_tw_reuse</code>	<p>This parameter is used to reuse a socket in TIME-WAIT state for new TCP connections. This help you quickly restart links that use fixed ports, but may involve risks in the NAT-based network. Later kernel versions support values 0, 1, and 2 and configure it to 2.</p>	-
<code>net.ipv4.ip_forward</code> <code>net.ipv6.conf.all.forwarding</code>	<p>This parameter is used to specify the IP forwarding. You can configure it to 1 in the Docker's route forwarding scenario.</p>	0
<code>net.ipv4.conf.default.rp_filter</code>	<p>This parameter is used to specify the reverse path validation rule of ENI on received data packets. Valid values include 0, 1 (recommended by RFC3704), and 2. The recommended configuration is a strict mode that can prevent DDoS attacks and IP spoofing acts.</p>	-
<code>net.ipv4.conf.default.accept_source_route</code>	<p>This parameter is used to specify</p>	0

	whether to accept IP packets containing source routes, which is not allowed by default, as recommended on the CentOS website.	
<pre>net.ipv4.conf.all.promote_secondaries</pre> <pre>net.ipv4.conf.default.promote_secondaries</pre>	This parameter is used to specify whether a secondary IP address will become a primary IP after the original primary IP address is deleted.	1
<pre>net.ipv6.neigh.default.gc_thresh3</pre> <pre>net.ipv4.neigh.default.gc_thresh3</pre>	This parameter is used to define the maximum records stored in the ARP cache. The garbage collector immediately starts once the stored records exceed the set value.	4096

Memory-related

Parameter	Description	Initial Configuration
<pre>vm.vfs_cache_pressure</pre>	This controls the tendency of the kernel to reclaim the memory. At the default value of 100, the kernel will attempt to reclaim dentries back to memory. The curl-based services usually accumulate	250

	<p>dentries, which may use up all the free memory and cause OOM or kernel bug. We configure it to 250 to balance the reclaiming frequency and performance, which is tunable.</p>	
<code>vm.min_free_kbytes</code>	<p>This parameter is used to force the Linux MEM to keep a minimum number of kilobytes free memory for use by kernel threads. The value is automatically calculated according to the free physical memory (MEM) at startup by: $4 * \sqrt{\text{MEM}}$ (MEM). When the server receives microbursts of packets, your server may become subtly broken and cause OOM. On a high-configuration server, we recommend configuring <code>vm.min_free_kbytes</code> at about 1% of the total memory by default.</p>	-
<code>kernel.printk</code>	<p>This specifies the level for kernel's <code>printk</code> printing function. The</p>	5 4 1 7

	default configuration is five or above.	
<code>kernel.numa_balancing</code>	This indicates that kernel can automatically move processes to the corresponding NUMA node, but it is actually ineffective and affects performance. You can try to enable it in the Redis use cases.	0
<code>kernel.shmallkernel.shmmax</code>	shmmax: defines the maximum size of a single shared memory segment (in bytes) a Linux process can allocate. shmall: defines system-wide total amount of shared memory pages.	<code>kernel.shmmax=68719476736kernel.shmall=42</code>

Process-related

Parameter	Description	Initial Configuration
<code>fs.file-maxfs.nr_open</code>	They denote the maximum number of file-handles that the Linux kernel or a process can allocate, respectively. file-max: automatically configures to approximate 100,000/GB when OS starts. nr_open: sets to the fixed value of 1048576, which limits the maximum open file handles in a user-mode environment. Generally, keep this value	<code>ulimit -n=100001fs.nr_open=1048576</code>

	unchanged. To modify the maximum open file handles, configure the ulimit -n parameter in the /etc/security/limits.conf configuration file.	
<code>kernel.pid_max</code>	This specifies maximum processes in a system. The official image uses the default value of 32768, which can be adjusted as needed.	-
<code>kernel.core_uses_pid</code>	This determines whether the generated coredump filename will contain .PID.	1
<code>kernel.sysrq</code>	This enables you to operate on /proc/sysrq-trigger later.	1
<code>kernel.msgmnb</code> <code>kernel.msgmax</code>	They defines the maximum size in bytes of a single message queue and the maximum allowable size in bytes of any single message in a message queue, respectively	65536
<code>kernel.softlockup_panic</code>	This controls whether the kernel will panic when a soft lockup is detected. If enabled, a vmcore will be generated based on the kdump configuration, which can be used to analyze the cause of soft lockup.	-

IO-related

Parameter	Description	Initial Configuration
<code>vm.dirty_background_bytes</code> <code>vm.dirty_background_ratio</code> <code>vm.dirty_bytes</code> <code>vm.dirty_expire_centisecs</code> <code>vm.dirty_ratio</code> <code>vm.dirty_writeback_centisecs</code>	These parameters are mainly used to configure the policy for IO being written back to the disk. <code>dirty_background_bytes/dirty_bytes</code> and <code>dirty_background_ratio/dirty_ratio</code> refer to the amount and percentage of system memory that can be filled with “dirty” pages, respectively. In general, the ratio will be specified.	-

dirty_background_ratio: refers to a percentage of dirty pages in the system memory (10% by default) at which the background kernel flush processes will start writing back to the disk.

dirty_ratio: refers to the absolute maximum amount of system memory that can be filled with dirty pages before everything must get committed to disk. When the system gets to this point, all new I/O blocks until dirty pages have been written to disk, causing long I/O pauses. The system will first get to the `vm.dirty_background_ratio` condition at which the flush processes will start asynchronous writeback, and applications continue writing. When the system gets to the specified value of `vm.dirty_ratio`, OS will handle dirty pages synchronously, blocking applications.

vm.dirty_expire_centisecs: specifies how long dirty page can be in cache before it needs to be written. It is expressed in 100'ths of a second. Data which has been dirty in-memory for longer than this interval will be written out next time a flush process wakes up.

vm.dirty_writeback_centisecs: specifies how often kernel flush processes wake up. It is expressed in 100'ths of a second.

Others

Booting into Linux Single User Mode

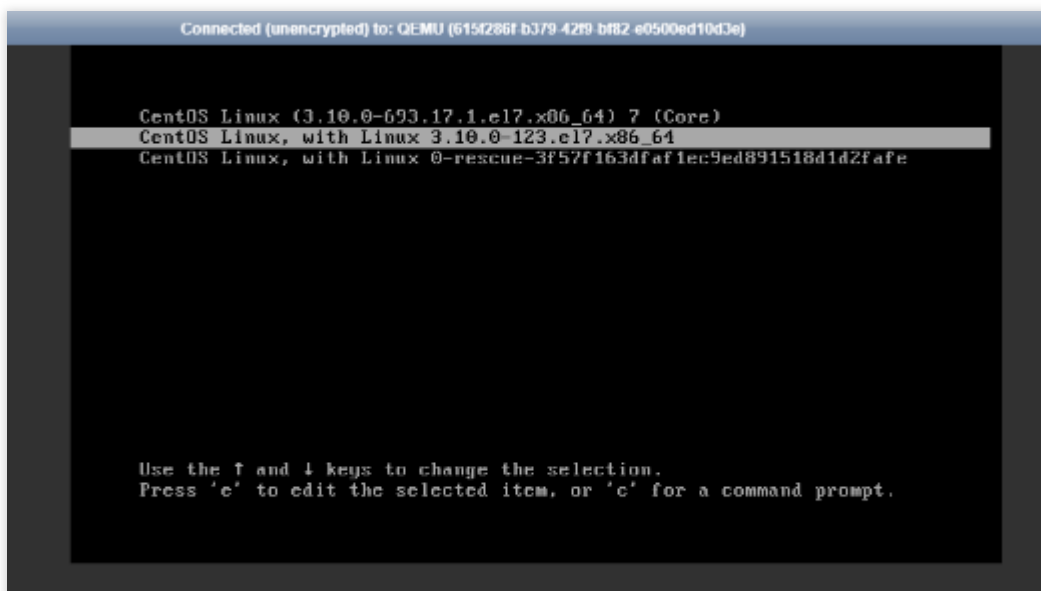
Last updated : 2024-01-06 17:38:11

Overview

For operations like password management, SSHD fixing and maintenance before disk mounting, Linux users need to enter the single user mode. This document describes how to boot into this mode in mainstream Linux distributions.

Directions

1. On the CVM console, choose VNC to log in to a CVM. For detailed directions, see [Logging into Linux Instances via VNC](#).
2. In the VNC login window, select **Send CtrlAltDel** in the upper-left corner, press **Ctrl-Alt-Delete**, and click **OK** in the prompt box.
3. When a connection failure message appears, press Up or Down arrow to refresh the page and hovers the cursor over the `grub` menu, as shown below.



4. Press **e** to enter the grub rescue mode.
5. Perform the steps that suit your operating system version.

CentOS 6.x

CentOS 7.x

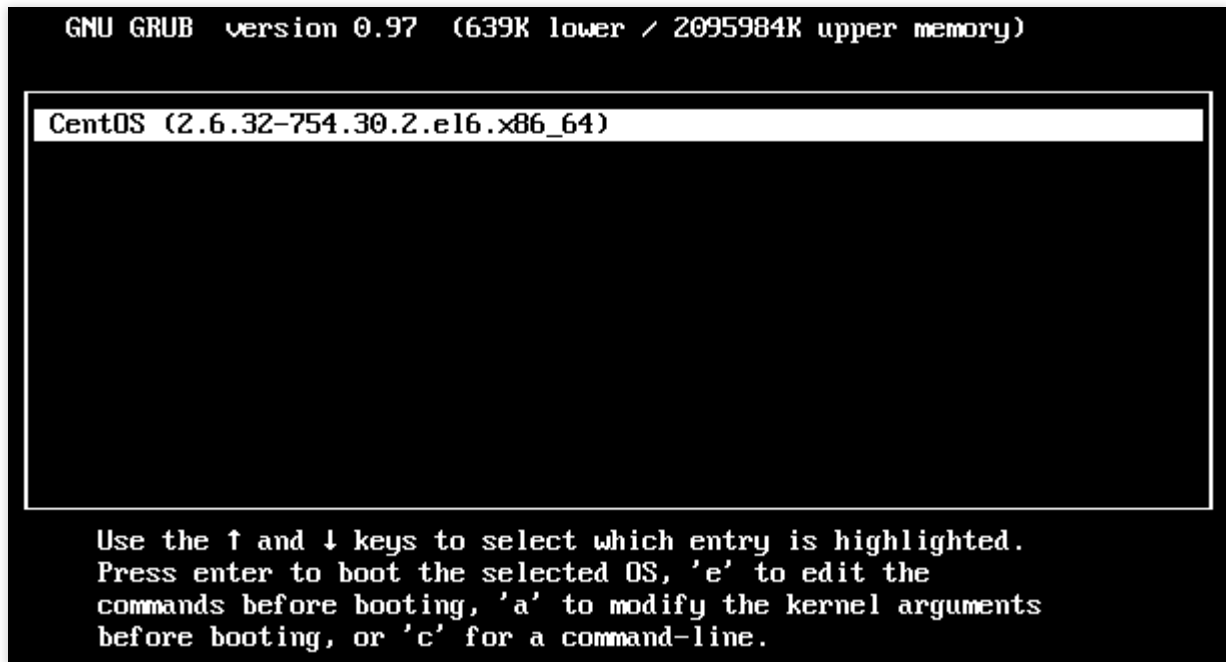
CentOS 8.0

Ubuntu or Debian

SUSE

tlinux

1. Select a kernel in the grub mode, as shown below.



2. Press **e** to enter the kernel edit page, choose the **kernel** line using Up or Down arrow, and press **e** again, as shown below.

```
GNU GRUB  version 0.97  (639K lower / 2095984K upper memory)

root (hd0,0)
kernel /boot/vmlinuz-2.6.32-754.30.2.el6.x86_64 ro root=/dev/vda1 con=
initrd /boot/initramfs-2.6.32-754.30.2.el6.x86_64.img

Use the ↑ and ↓ keys to select which entry is highlighted.
Press 'b' to boot, 'e' to edit the selected command in the
boot sequence, 'c' for a command-line, 'o' to open a new line
after ('O' for before) the selected line, 'd' to remove the
selected line, or escape to go back to the main menu.
```

3. Enter **single** at the end of line, as shown below.

```
[ Minimal BASH-like line editing is supported. For the first word, TAB
lists possible command completions. Anywhere else TAB lists the possible
completions of a device/filename. ESC at any time cancels. ENTER
at any time accepts your changes.]

<l_pstate=disable single|
```

4. Press **Enter**, and then press **b** to boot the selected command line and enter single user mode, as shown below.

```
GNU GRUB version 0.97 (639K lower / 2095984K upper memory)

root (hd0,0)
kernel /boot/vmlinuz-2.6.32-754.30.2.el6.x86_64 ro root=/dev/vda1 con=
initrd /boot/initramfs-2.6.32-754.30.2.el6.x86_64.img

Use the ↑ and ↓ keys to select which entry is highlighted.
Press 'b' to boot, 'e' to edit the selected command in the
boot sequence, 'c' for a command-line, 'o' to open a new line
after ('O' for before) the selected line, 'd' to remove the
selected line, or escape to go back to the main menu.
```

The following figure indicates that the system boots into single user mode.

```
                Welcome to CentOS
Starting udev: [ OK ]
Setting hostname VM-1-111-centos: [ OK ]
Setting up Logical Volume Management: [ OK ]
Checking filesystems
/dev/vda1: clean, 32886/3276800 files, 508979/13106775 blocks
Remounting root filesystem in read-write mode: [ OK ]
Mounting local filesystems: [ OK ]
Enabling local filesystem quotas: [ OK ]
Enabling /etc/fstab swaps: [ OK ]
[root@VM-1-111-centos /]# _
```

Note:

You can run the `exec /sbin/init` command to exit the single user mode.

1. Select a kernel in the grub mode, as shown below.

```
CentOS Linux (3.10.0-1127.19.1.el7.x86_64) 7 (Core)
```

```
Use the ↑ and ↓ keys to change the selection.
Press 'e' to edit the selected item, or 'c' for a command prompt.
The selected entry will be started automatically in 1s.
```

2. Press **e** to enter the kernel edit page. Locate the line started with “linux16” using Up or Down arrow and replace **ro** with **rw init=/bin/bash** or **/usr/bin/bash**, as shown below.

```
insmod part_msdos
insmod ext2
set root='hd0,msdos1'
if [ x$feature_platform_search_hint = xy ]; then
    search --no-floppy --fs-uuid --set=root --hint='hd0,msdos1' 21dbe0
0-aa71-4b3a-8610-3b942dd447fa
else
    search --no-floppy --fs-uuid --set=root 21dbe030-aa71-4b3a-8610-3b9
2dd447fa
fi
linux16 /boot/vmlinuz-3.10.0-1127.19.1.el7.x86_64 root=UUID=21dbe030-
a71-4b3a-8610-3b942dd447fa rw init=/bin/bash crashkernel=auto console=ttyS0 c
nsole=tty0 panic=5 net.ifnames=0 biosdevname=0 intel_idle.max_cstate=1 intel_
state=disable
initrd16 /boot/initramfs-3.10.0-1127.19.1.el7.x86_64.img

Press Ctrl-x to start, Ctrl-c for a command prompt or Escape to
discard edits and return to the menu. Pressing Tab lists
possible completions.
```

3. Press **Ctrl+X** to boot into the single user mode, as shown below:

The following figure indicates that the system boots into the single user mode.

```
[ OK ] Stopped Create list of required static device nodes for the current kernel
[ OK ] Closed udev Kernel Socket.
[ OK ] Closed udev Control Socket.
        Starting Cleanup udevd DB...
[ OK ] Started Cleanup udevd DB.
[ OK ] Reached target Switch Root.
[ OK ] Started Plymouth switch root service.
        Starting Switch Root...
bash-4.2# _
```

Note:

You can run the `exec /sbin/init` command to exit the single user mode.

1. Select a kernel in the grub mode, as shown below.

```
CentOS Linux (4.18.0-80.el8.x86_64) 8 (Core)
CentOS Linux (0-rescue-33790f3e0323419f9a055840e9d10b13) 8 (Core)

Use the ↑ and ↓ keys to change the selection.
Press 'e' to edit the selected item, or 'c' for a command prompt.
```

2. Press **e** to enter the kernel edit page. Locate the line started with “linux” using Up or Down arrow and replace `ro` with `rw init=/sysroot/bin/bash` , as shown below.

```

insmod ext2
set root='hd0,msdos1'
if [ x$feature_platform_search_hint = xy ] then
    search --no-floppy --fs-uuid --set=root --hint='hd0,msdos1' 659e6f89-71fa-463d-842e-ccdf2c06e0fe
else
    search --no-floppy --fs-uuid --set=root 659e6f89-71fa-463d-842e-ccdf2c06e0fe
fi
linux      /boot/vmlinuz-4.18.0-80.el8.x86_64 root=UUID=659e6f89-71fa-463d-842e-ccdf2c06e0fe rw init=/sysroot/bin/sh crashkernel=auto console=tty0 console=tty0 panic=5 net.ifnames=0 biosdevname=0 intel_idle.max_cstate=1 intel_pstate=disable
initrd     /boot/initramfs-4.18.0-80.el8.x86_64.img

Press Ctrl-x to start, Ctrl-c for a command prompt or Escape to
discard edits and return to the menu. Pressing Tab lists
possible completions.

```

3. Press **Ctrl+X** to boot into the single user mode, as shown below:

The following figure indicates that the system boots into the single user mode.

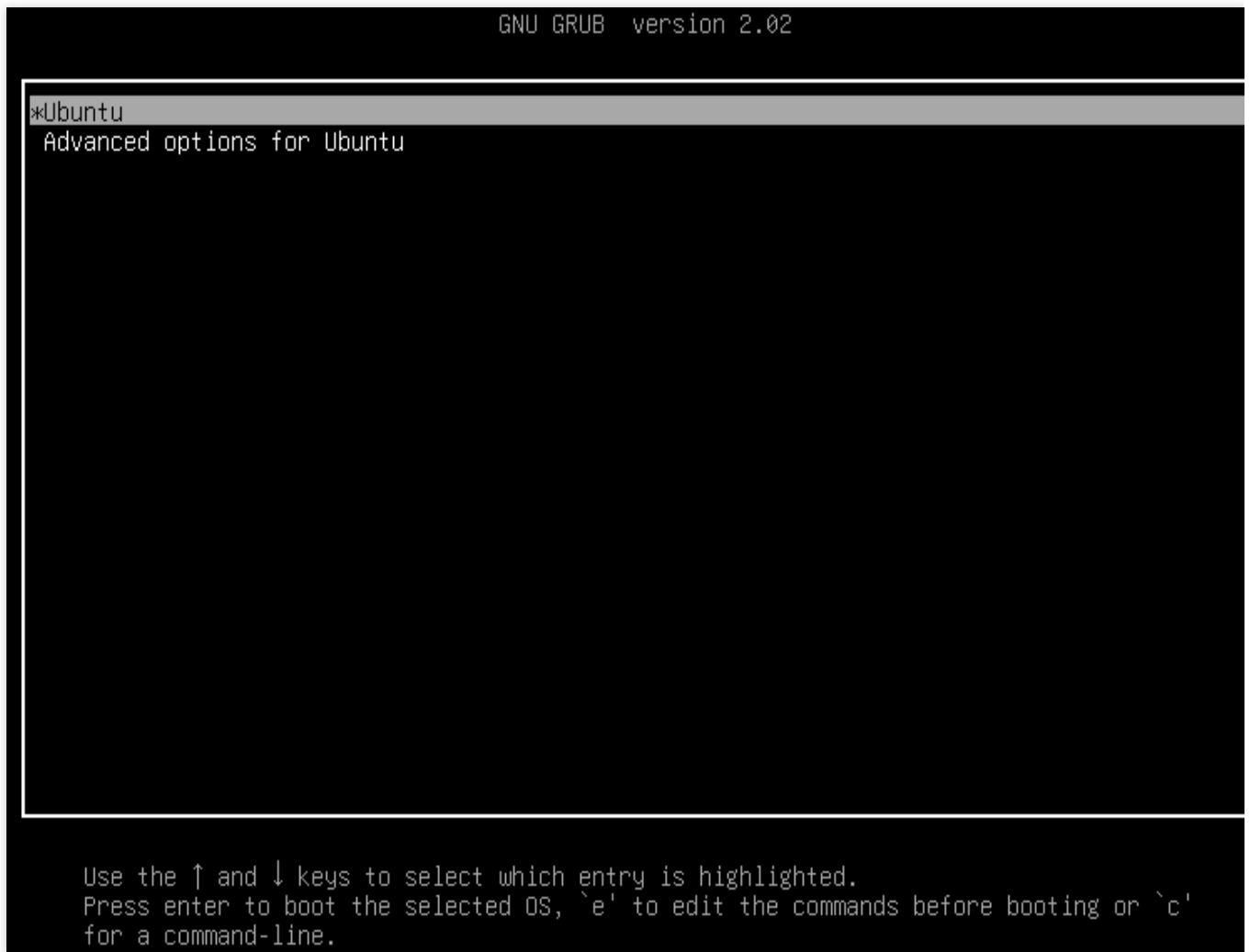
```

Entering emergency mode. Exit the shell to continue.
Type "journalctl" to view system logs.
You might want to save "/run/initramfs/rdsosreport.txt" to a USB stick or /boot
after mounting them and attach it to a bug report.

:/# _

```

1. Select a kernel in the grub mode, as shown below.



2. Press **e** to enter the kernel edit page. Locate the line started with "linux" using Up or Down arrow and append `quiet splash rw init=/bin/bash` to the end of the line, as shown below.

```

GNU GRUB version 2.02

setparams 'Ubuntu'

    recordfail
    load_video
    gfxmode $linux_gfx_mode
    insmod gzio
    if [ x$grub_platform = xxen ]; then insmod xzio; insmod lzopio; fi
    insmod part_msdos
    insmod ext2
    if [ x$feature_platform_search_hint = xy ]; then
        search --no-floppy --fs-uuid --set=root 5ba34c3d-bd14-451d-a7d8-09a64009e3f1
    else
        search --no-floppy --fs-uuid --set=root 5ba34c3d-bd14-451d-a7d8-09a64009e3f1
    fi
    linux      /boot/vmlinuz-4.15.0-118-generic root=UUID=5ba34c3d-bd14-451d-a7d8-09a
4009e3f1 ro net.ifnames=0 biosdevname=0 console=ttyS0,115200 console=tty0 panic=5 intel_idl
.max_cstate=1 intel_pstate=disable crashkernel=1800M-64G:160M,64G-:512M quiet splash rw in
t=/bin/bash
    initrd     /boot/initrd.img-4.15.0-118-generic

Minimum Emacs-like screen editing is supported. TAB lists completions. Press Ctrl-x
or F10 to boot, Ctrl-c or F2 for a command-line or ESC to discard edits and return
to the GRUB menu.

```

3. Press **Ctrl+X** to boot into the single user mode, as shown below:

The following figure indicates that the system boots into single user mode.

```

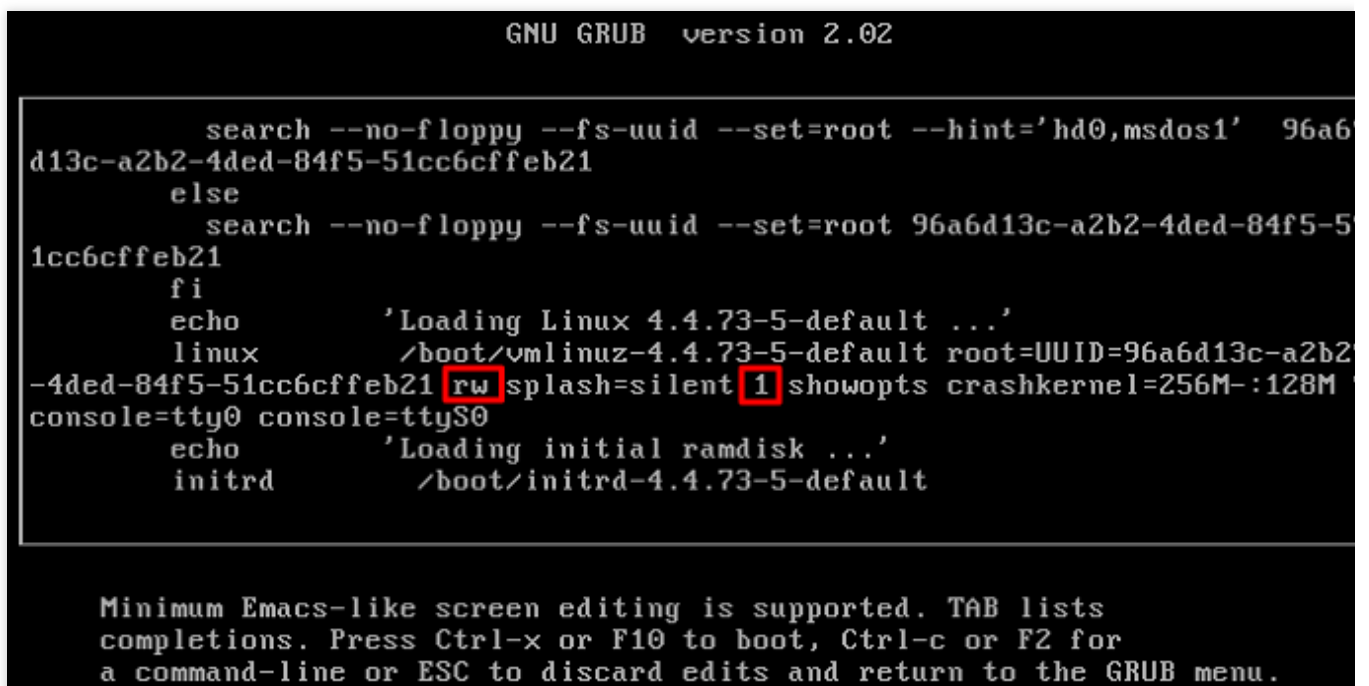
/dev/vda1: clean, 83224/1310720 files, 669241/5242619 blocks
bash: cannot set terminal process group (-1): Inappropriate ioctl for device
bash: no job control in this shell
root@(none):/#

```

1. Select a kernel in the grub mode, as shown below.

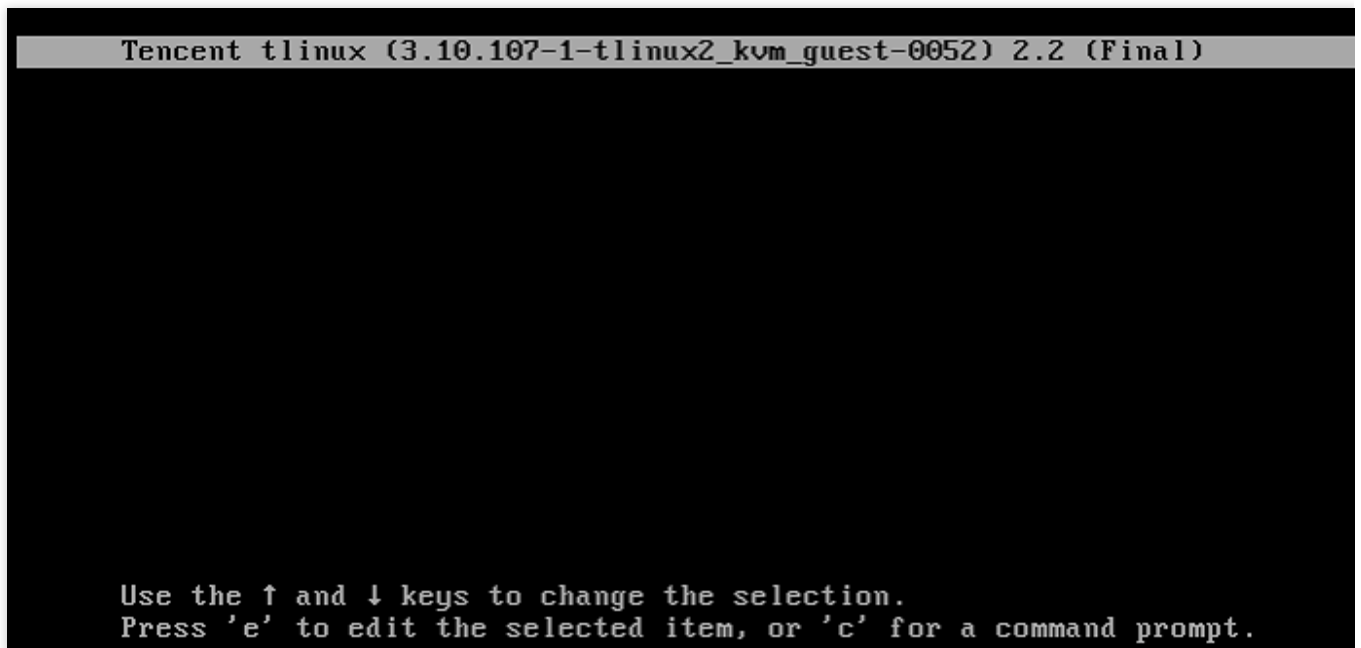


2. Press **e** to enter the kernel edit page. Locate the line started with "linux" using Up or Down arrow, add **rw** to the beginning and **1** to the end of the **splash** parameter, as shown below.

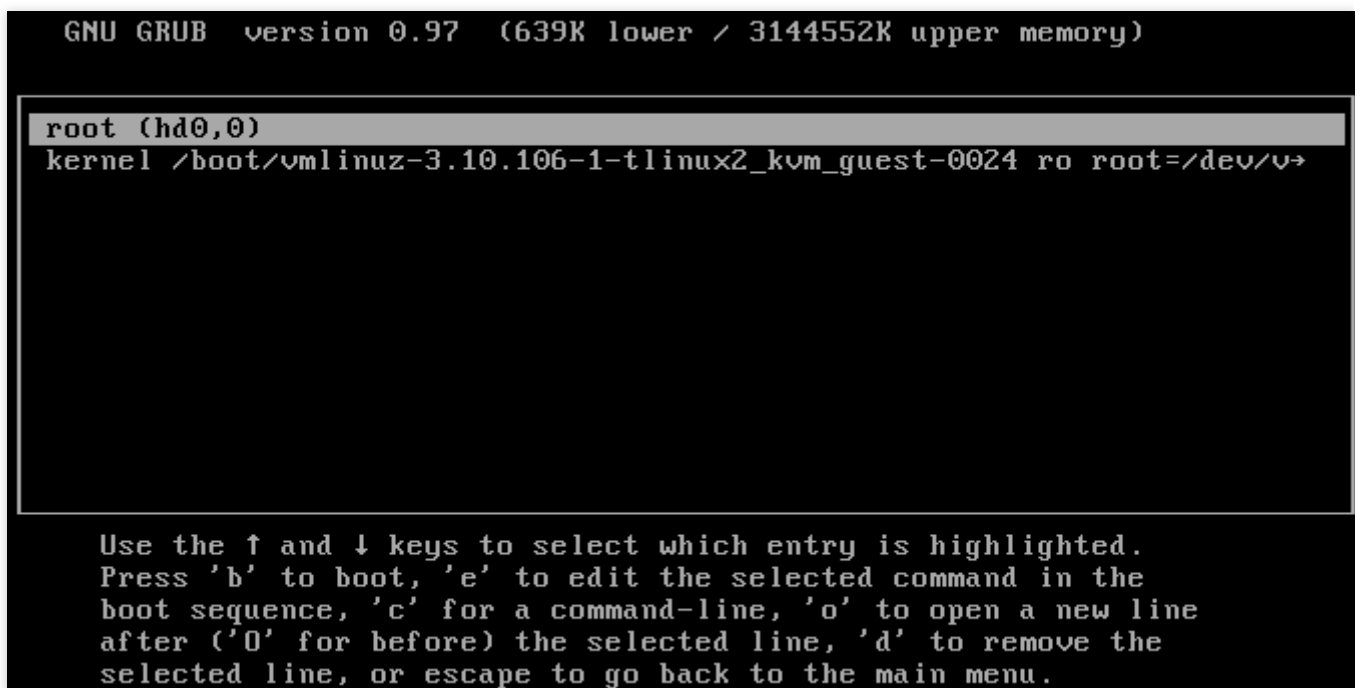


3. Press **Ctrl+X** to boot into the single user mode, as shown below:

1. Select a kernel in the grub mode, as shown below.



2. Press **e** to enter the kernel edit page, choose the **kernel** line using Up or Down arrow, and press **e** again, as shown below.



3. Add a space and **1** to the end of the line (namely after **256M**), as shown below.

```
Connected (unencrypted) to: QEMU (612bab60-5090-4b1d-a19f-6dae578cc8e4)
[ Minimal BASH-like line editing is supported. For the first word, TAB
  lists possible command completions. Anywhere else TAB lists the possible
  completions of a device/filename. ESC at any time cancels. ENTER
  at any time accepts your changes.]

<8M,12G-:256M
```

4. Press **Enter** to enter the single user mode.

Using Rescue Mode

Last updated : 2024-01-06 17:38:11

Overview

This document describes how to use the rescue mode in the CVM console. If the server GRUB bootloader files, key system files, or .lib dynamic library files are corrupted or missing when you use the CVM operating system, the operating system may be unable to enter the single user mode and be repaired. In this case, you need to use the CVM rescue mode to repair the system.

Directions

Entering rescue mode

Note:

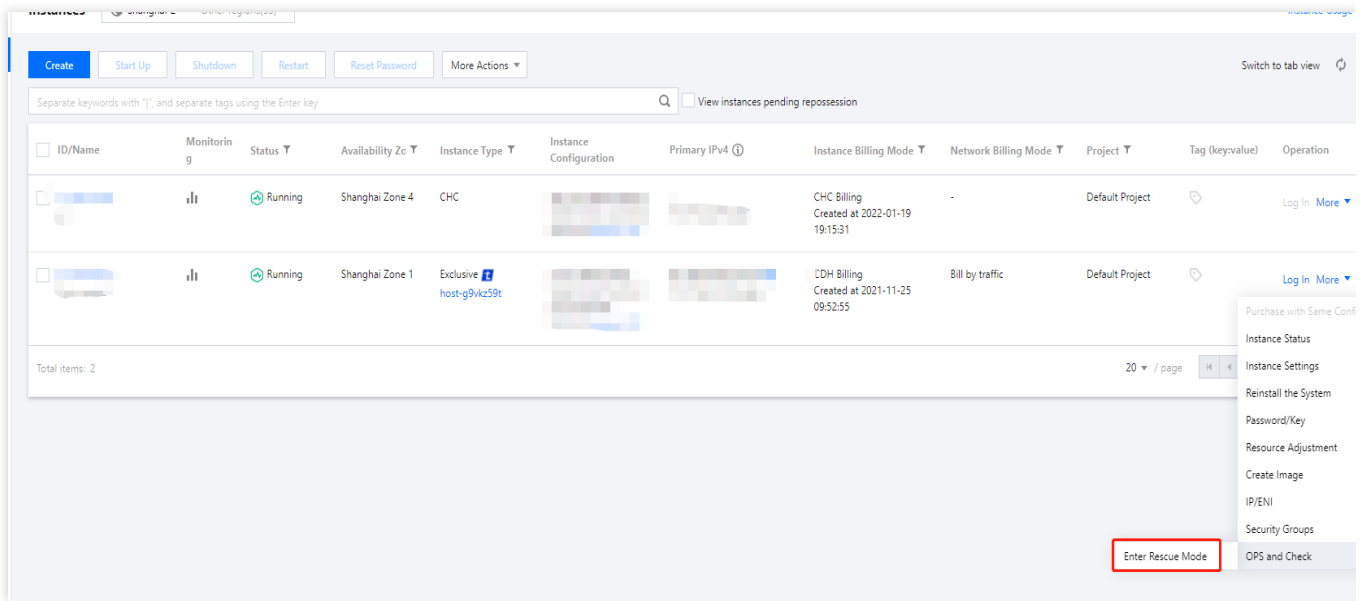
Before you enter the rescue mode, we strongly recommend you back up the instance to avoid the impact of maloperations. You can [create snapshots](#) to back up cloud disks and [create custom images](#) to back up local system disks.

1. Log in to the [CVM console](#).
2. On the instance management page, proceed according to the used view mode:

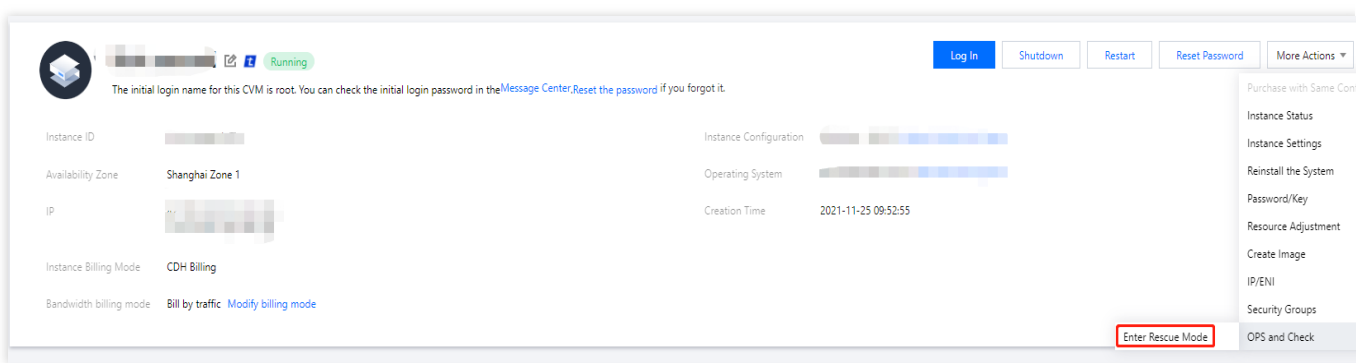
List mode

Tab mode

Select **More > OPS and Check > Enter Rescue Mode** on the right of the row of the target instance as shown below:



Click the tab of the target instance and select **More Actions > OPS and Check > Enter Rescue Mode** in the top-right corner as shown below:



3.

In the **Enter Rescue Mode** pop-up window, set the instance login password for the rescue mode as shown below:

Note:

Currently, the rescue mode supports only Linux instances but not Windows instances. If you enter the rescue mode in a Windows instance, you will enter the Linux rescue mode (on CentOS 7.5 64-bit) by default.

The instance username is `root` by default in the rescue mode.

You can enter the rescue mode only when the instance is shut down. Forced shutdown may cause data loss or file system corruption. We recommend you shut down the instance as instructed in [Shutting Down Instance](#) before entering the rescue mode.

Enter Rescue Mode ✕

i 1. Before entering the rescue mode, you need to set a password, which is used to access the instance during the rescue period. The default username is "root". After exiting the Rescue Mode, you need to access the instance with the original password.

2. In the Rescue Mode, the instance starts up from CD-ROM by default. The operating system for CD-ROM start-up is CentOS 7.5 64-bit.

3. When an instance is in Rescue Mode, it cannot be started up or shut down.

4. To enter the Rescue Mode, the instance should be shut down. Forced shutdown may result in data loss or file system corruption. We recommend manually shutting down the CVM manually before the operation.

5. After exiting the Rescue Mode, the CVM instance will be "shut down" by default. Please immediately restart it.

Password

The password for a Linux server should be a combination of 8 to 30 characters. The password cannot start with "/", and must include at least 3 types of the following characters: [a-z], [A-Z], [0-9] and special characters [()~!@#\$%^&*~+=_{}|;:'<>.,?/]

Please enter the rescue mode access password.

Confirm Password

Please enter the password again.

Forced Shutdown ☐ Agree to a forced shutdown

Forced shutdown may take a while. Please be patient.

Enter Rescue Mode

Close

4. Click **Enter Rescue Mode**.

At this point, you can see that the instance is entering the rescue mode. If the instance status is as shown below, the instance has entered the rescue mode successfully. Proceed to the next step to repair the instance as soon as possible.

Create

Start Up

Shutdown

Restart

Reset Password

More Actions

Switch to tab view

Separate keywords with " ", and separate tags using the Enter key

View instances pending reposition

<input type="checkbox"/>	ID/Name	Monitoring	Status	Availability Zone	Instance Type	Instance Configuration	Primary IPv4	Instance Billing Mode	Network Billing Mode	Project	Tag (key=value)	Operation
<input type="checkbox"/>			Running	Shanghai Zone 4	CHC			CHC Billing Created at 2022-01-19 19:15:31	-	Default Project		Log In More
<input type="checkbox"/>			Rescue Mode	Shanghai Zone 1	Exclusive host-g9vkc39t			CDH Billing Created at 2021-11-25 09:52:55	Bill by traffic	Default Project		Log In More

Total items: 2

20 / page

1

/ 1 page

Using rescue mode to repair system

1. Use the `root` account and the password set in [step 3](#) to log in to the instance as follows:

If the instance has an EIP, you can log in as instructed in [Logging into Linux Instance via SSH Key](#).

If the instance has no EIPs, you can log in as instructed in [Logging into Linux Instances via VNC](#).

2. After successful login, run the following commands in sequence to mount the root system disk partition:

In rescue mode, the device name of the instance system disk is `vda`, and its root partition is `vda1`, which is unmounted by default.

```
mkdir -p /mnt/vm1

mount /dev/vda1 /mnt/vm1
```

After successful mounting, you can manipulate the data in the root partition. You can also use the `mount -o bind` command to mount some sub-directories in the original file system and use the `chroot` command to run commands in the specified root directory. Below are the specific commands:

```
mount -o bind /dev /mnt/vm1/dev
mount -o bind /dev/pts /mnt/vm1/dev/pts
mount -o bind /proc /mnt/vm1/proc
mount -o bind /run /mnt/vm1/run
mount -o bind /sys /mnt/vm1/sys
chroot /mnt/vm1 /bin/bash
```

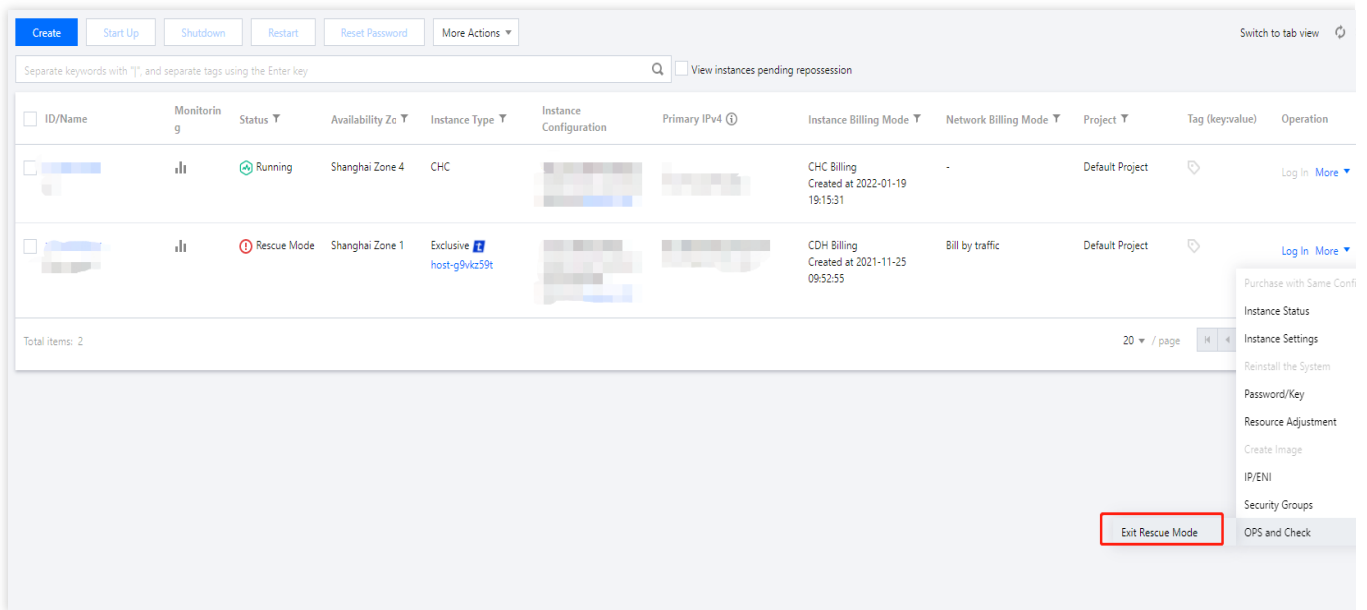
Exiting rescue mode

1. After repairing the instance, exit the rescue mode in the following steps according to the used view mode:

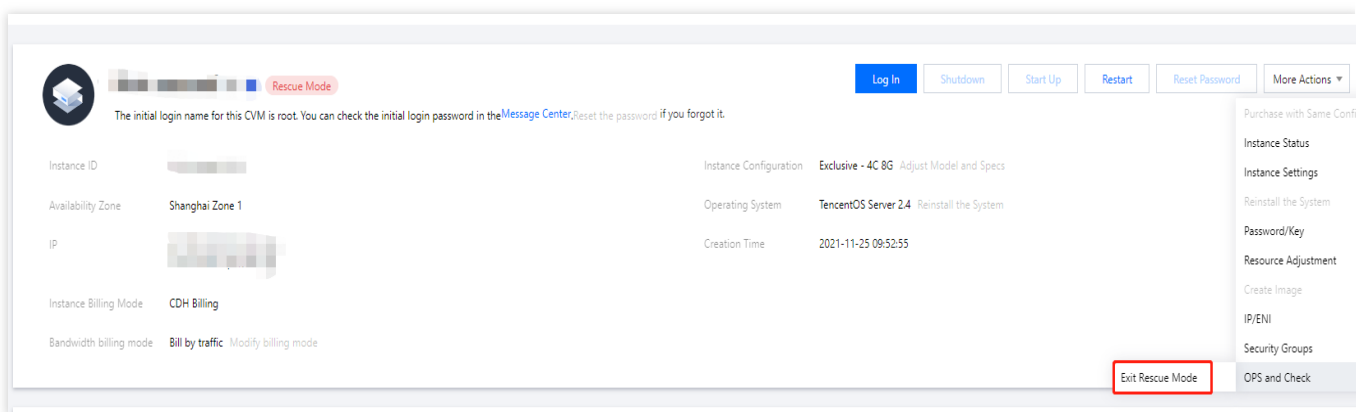
List mode

Tab mode

Select **More > OPS and Check > Exit Rescue Mode** on the right of the row of the target instance as shown below:



Click the tab of the target instance and select **More Actions > OPS and Check > Exit Rescue Mode** in the top-right corner as shown below:



2. After the instance exits the rescue mode, it is still shut down. Start it up as instructed in [Starting Up Instances](#) to resume instance operations.

Setting Up Windows CVM for Multi-user Remote Login

Last updated : 2024-01-06 17:38:11

Overview

This document shows you how to configure a multi-user remote login to Windows CVM, taking a CVM with Windows Server 2016 R2 as the operating system as an example.

Note:

The trial period of the multi-user remote login feature provided by Microsoft is 120 days. If you haven't purchased multi-user login licenses (RDS CALs), after the trial ends, you can log in to CVM only through the `mstsc /admin` command but not Remote Desktop. Windows Server allows two users to log in at the same time by default, which meets most needs. Evaluate your needs based on your actual business scenarios, and if you strongly need to configure multi-user remote login, proceed as instructed in this document.

Directions

Adding remote desktop service

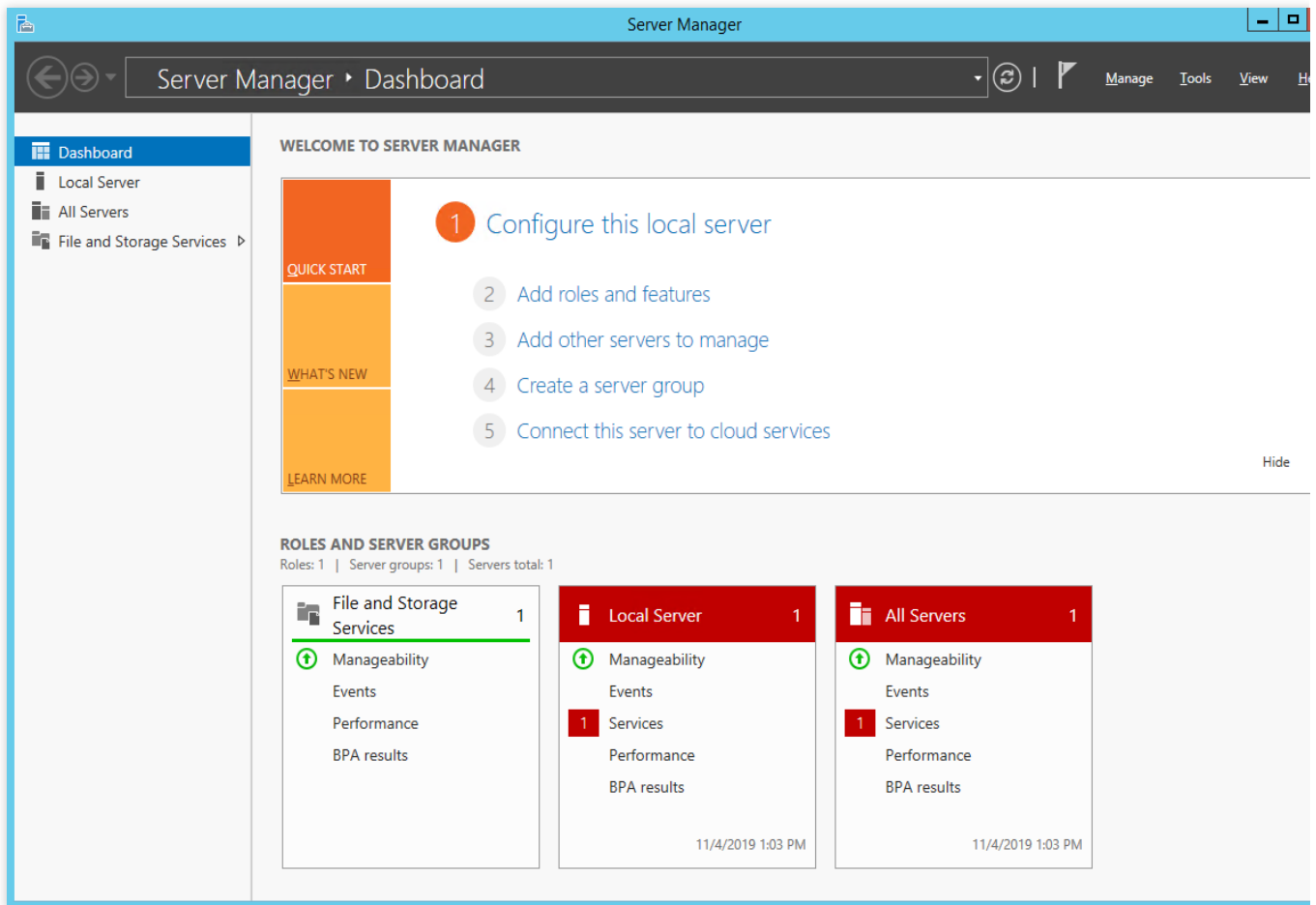
1. Log in to the Windows CVM.
2. On the desktop, click



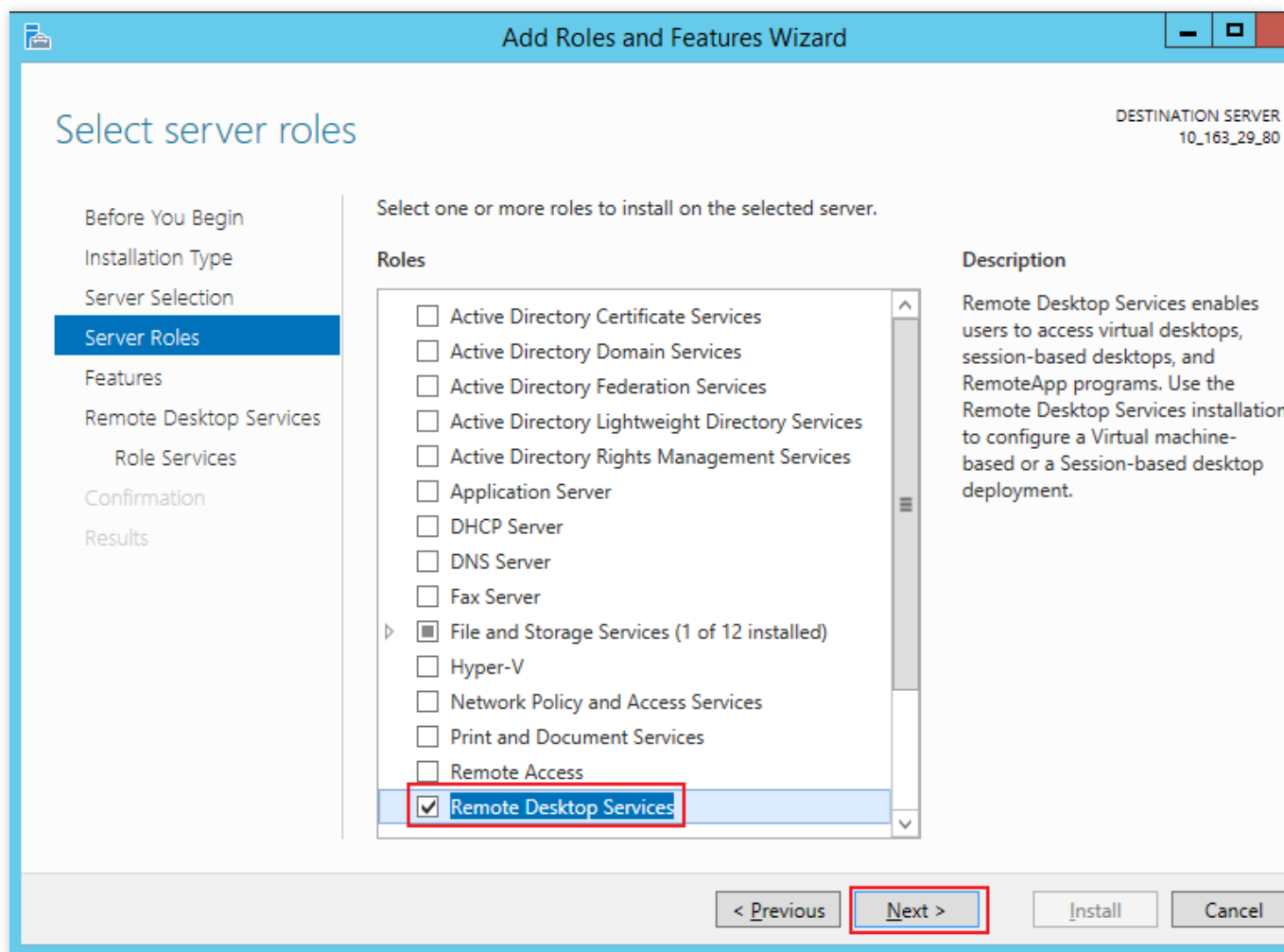
and select



in the pop-up window to open **Server Manager** as shown below:

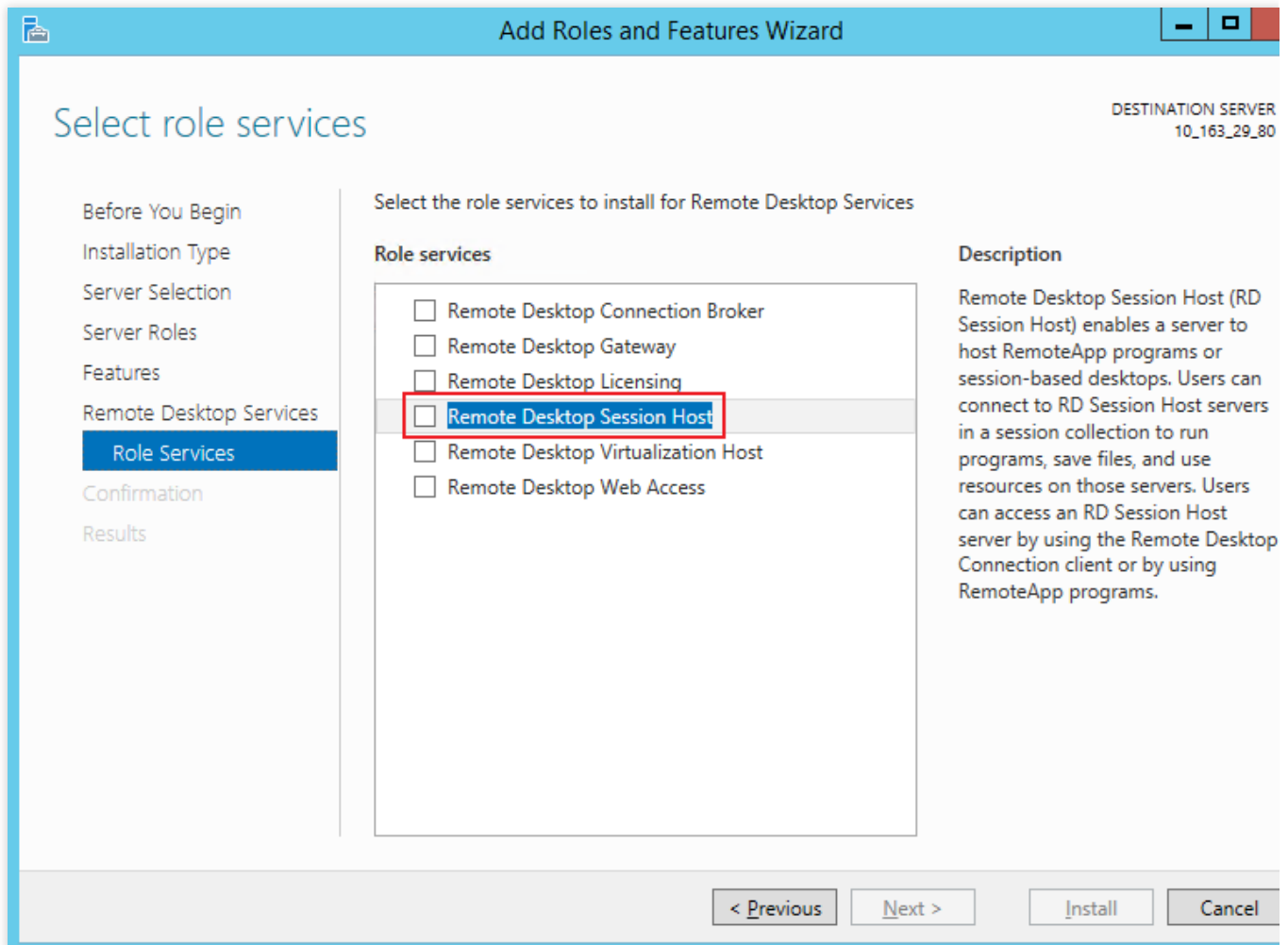


3. Click **Add roles and features**, and the **Add Roles and Features Wizard** will pop up.
4. In the **Add Roles and Features Wizard** window, keep the default parameters for the first three steps (click **Next** three times in a row).
5. On the **Select server roles** page, select **Remote Desktop Services** and click **Next** as shown below:

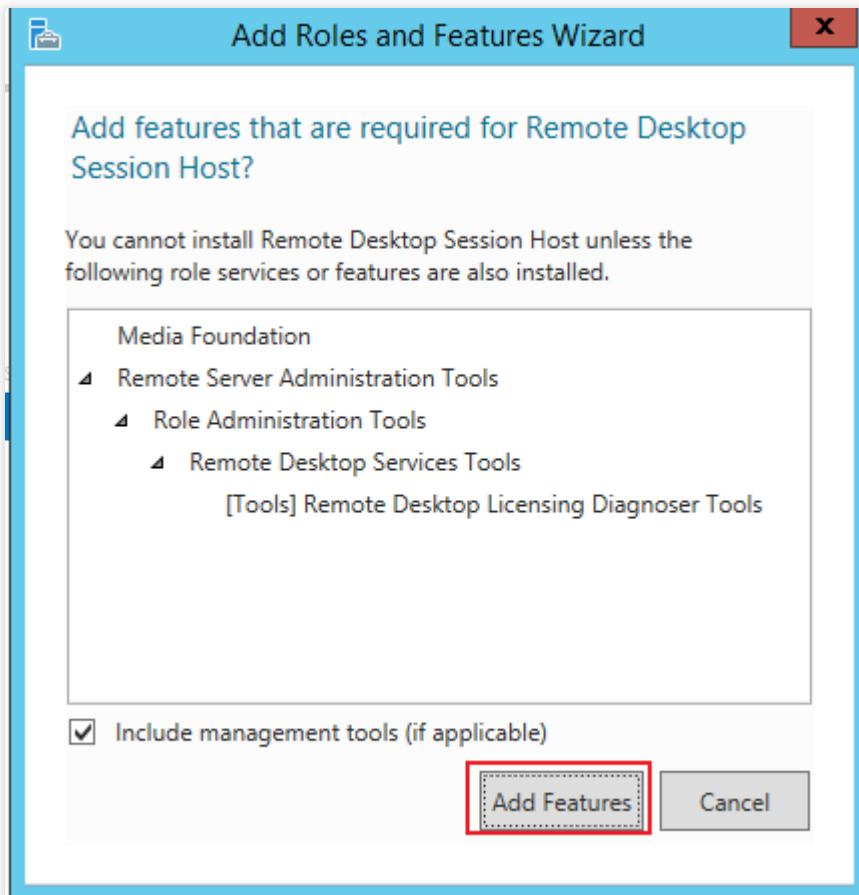


6. Keep the default parameters and click **Next** twice in a row.

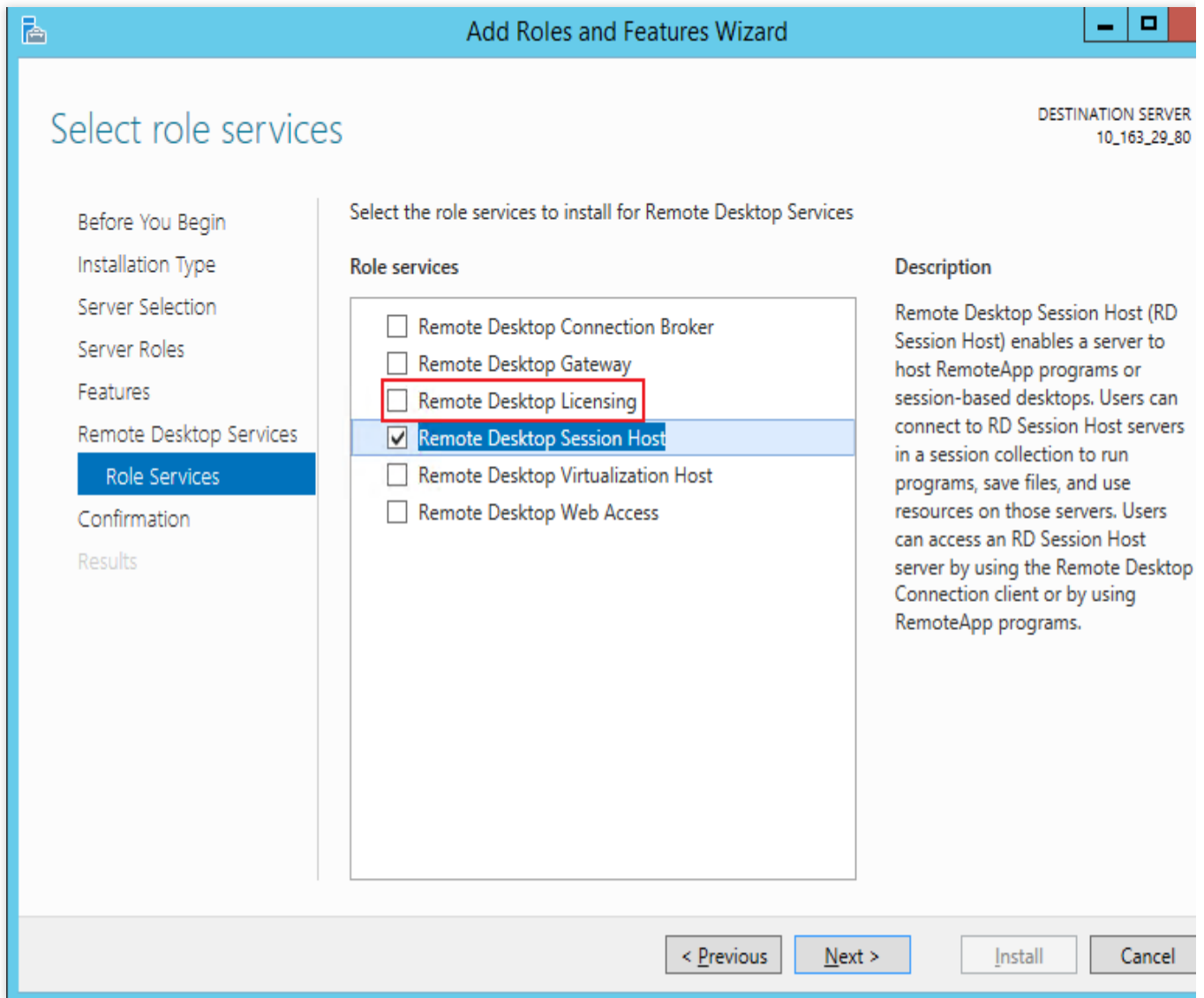
7. On the **Select role services** page, select **Remote Desktop Session Host** as shown below:



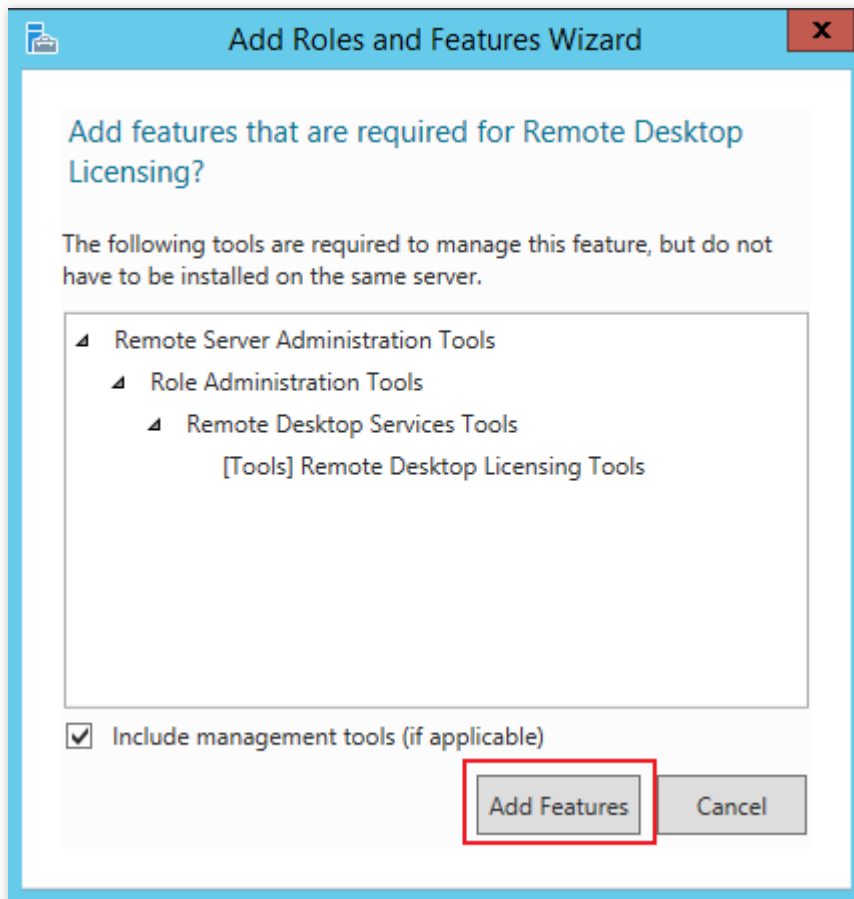
8. In the **Add features that are required for Remote Desktop Session Host?** pop-up window, click **Add Features** as shown below:



9. On the **Select role services** page, select **Remote Desktop Licensing** as shown below:

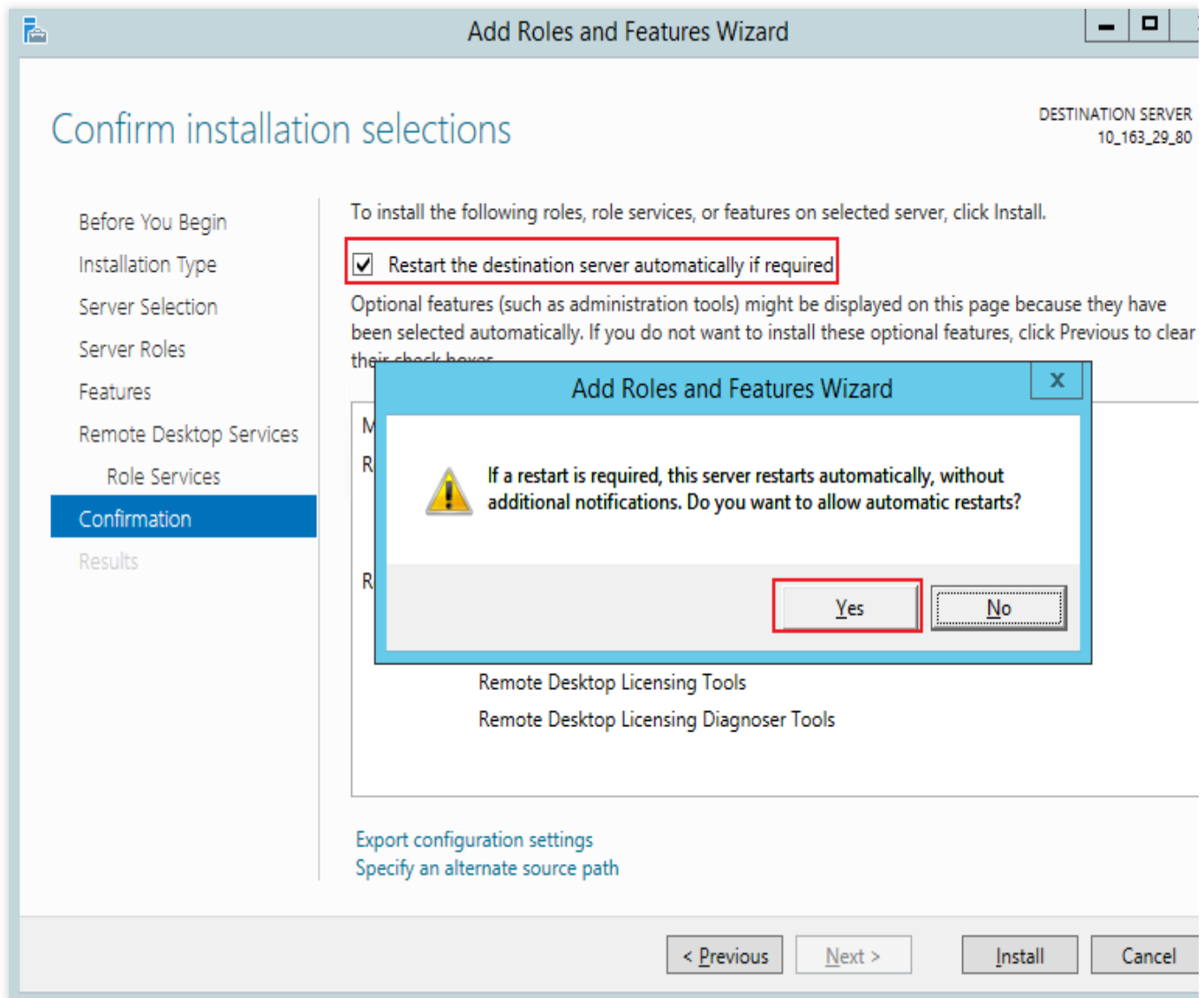


10. In the **Add features that are required for Remote Desktop Licensing?** pop-up window, click **Add Features** as shown below:



11. Click **Next**.

12. Select **Restart the destination server automatically if required** and click **Yes** in the pop-up window as shown below:



13. Click **Install** and wait for the Remote Desktop service installation to complete.

Applying for multi-user login license

1. On the desktop, click



and select



in the pop-up window to open **Server Manager**.

2. In the **Server Manager** window, select **Tools > Remote Desktop Services > Remote Desktop Licensing Manager** in the top-right corner.

3. In the **Remote Desktop Licensing Manager** pop-up window, right-click the row of the target server and select **Activate Server**.

4. In the **Activate Server Wizard** pop-up window, click **Next**.

5. Select **Web Browser** for **Connection method** and click **Next**.

You can also select other connection methods based on your actual conditions.

6. In the **License Server Activation**, record the product ID and visit the [Remote Desktop Licensing website](#).

7. At the Remote Desktop Licensing website, select **Activate a license server** and click **Next**.

8. Enter the product ID obtained in [step 6](#), enter the company information based on your actual conditions, and click **Next**.

9. After confirming that everything is correct, click **Next**.

10.

Record the license server ID

and click **Yes**.

11. Enter the license server ID obtained in the previous step, select the licensing information as needed, enter the company information, and click **Next**.

Here, **Enterprise agreement** is selected as the licensing information as an example.

12. Select the product type and enter the product quantity and licensing information.

Note:

You can go to the [Microsoft official website](#) and contact the customer service to purchase RDS CALs.

13. After confirming that everything is correct, click **Next**.

14.

Get and record the key pack ID

.

15. Click **Finish**.

Activating Remote Desktop license server

1. On the desktop, click



and select



in the pop-up window to open **Server Manager**.

2. In the **Server Manager** window, select **Tools > Remote Desktop Services > Remote Desktop Licensing Manager** in the top-right corner.

3. In the **Remote Desktop Licensing Manager** pop-up window, right-click the row of the target server and select **Activate Server**.

4. In the **Activate Server Wizard** pop-up window, click **Next**.

5. Select **Web Browser** for **Connection method** and click **Next**.

You can also select other connection methods based on your actual conditions.

6. In **License Server Activation**, enter the license server ID obtained in [step 10](#) and click **Next**.

7. When the **Activate Server Wizard** prompts **You have completed the Activate Server Wizard**, click **Next** to install the license.

Installing RDS client access license

1. In the **Install Licenses Wizard** window, confirm the license server information and click **Next**.

2. In **Obtain Client License Key Pack**, enter the license server ID obtained in [step 14](#) and click **Next**.

3. When the **Install Licenses Wizard** prompts **You have completed the Install Licenses Wizard** as shown below, you have installed the license successfully.

Configuring Remote Desktop Session Host license server

1. On the desktop, click



in the pop-up window to open **Server Manager**.

2. In the **Server Manager** window, select **Tools > Remote Desktop Services > Remote Desktop Licensing Diagnoser** and view the current server status.

3. On the desktop, right-click



and select **Run** in the pop-up window.

4. In the **Run** window, enter **gpedit.msc** and press **Enter** to open Local Group Policy Editor.

5. On the left sidebar, select **Computer Configuration > Administrative Templates > Windows Components > Remote Desktop Services > Remote Desktop Session Host > Licensing** and double-click **Use the specified Remote Desktop license servers**.

6. In the **Use the specified Remote Desktop license servers** pop-up window, select **Enabled**, enter a CVM instance public IP or host name for **License Servers to use**, and click **OK**.

7. Double-Click **Set the Remote Desktop licensing mode**.

8. In the **Set the Remote Desktop licensing mode** pop-up window, select **Enabled**, set the licensing mode for the Remote Desktop Session Host server to **By user**, and click **OK**.

9. Restart the CVM instance.

At this point, you have completed the configuration of multi-user remote login.

References

[License your RDS deployment with client access licenses \(CALs\)](#)

[Activate the Remote Desktop Services license server](#)

[Install RDS client access licenses on the Remote Desktop license server](#)

Modifying the Default Remote Port of CVM

Last updated : 2024-01-06 17:38:11

Scenario

The default port of the CVM is susceptible to scanning and attack by malicious software. Therefore, you need to change the default remote port of the CVM to a less common port to prevent the inability to remotely access the CVM due to such attacks. This ensures the security of the CVM.

Modifications to the port will be valid only if they are made in the security group rules and the CVM simultaneously. You can modify the default remote port of the CVM as described below. The modification method varies based on the operating system of the CVM.

[Modifying the default remote port of a Windows CVM](#)

[Modifying the default remote port of a Linux CVM](#)

Directions

Modifying the default remote port of a Windows CVM

Note:

The following operations use Windows Server 2012 as an example. The procedure might vary slightly depending on the operating system and language.

1. [Log in to the Windows instance by using VNC.](#)
2. In the operating system, click

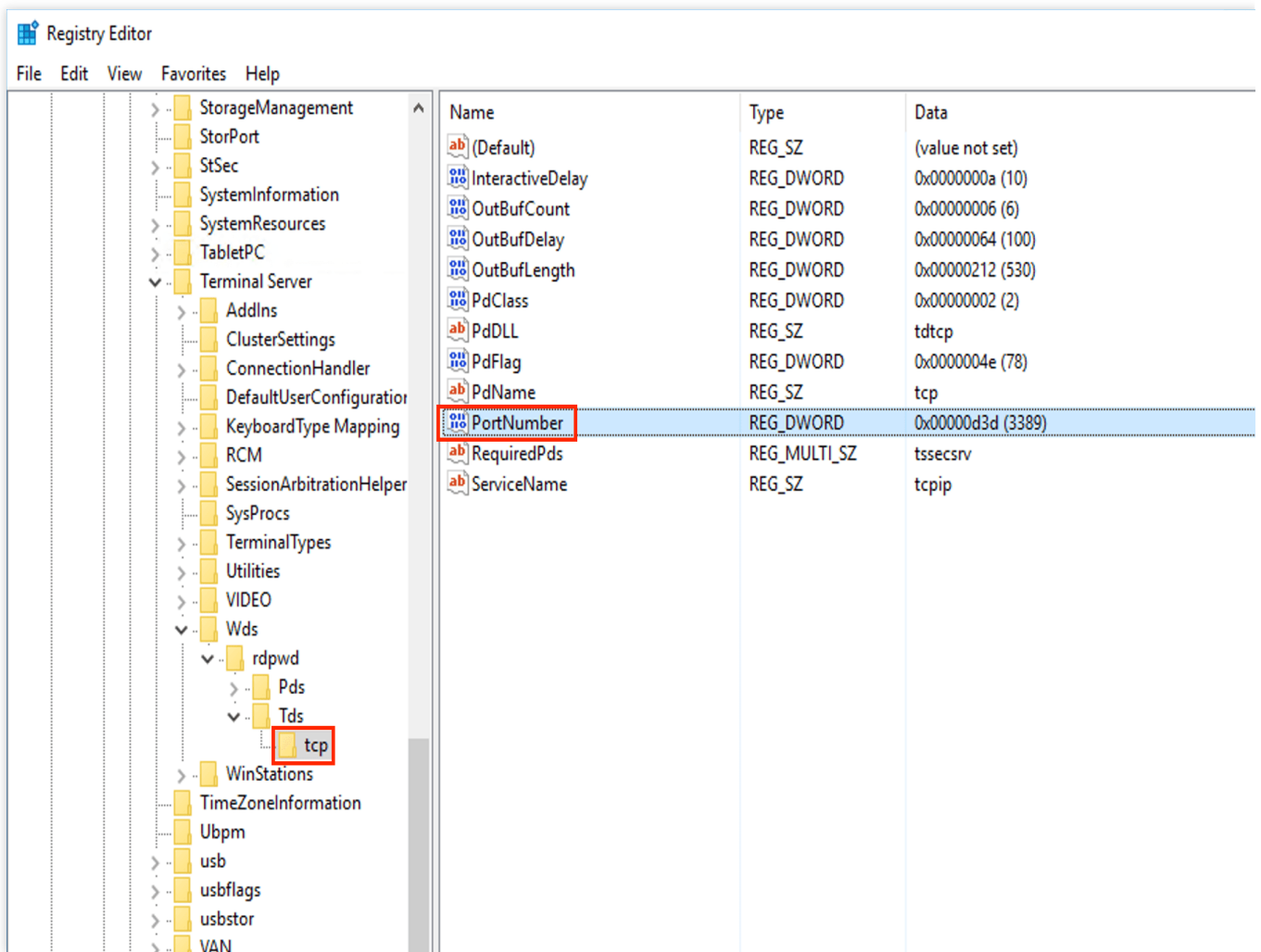


to open the "Windows PowerShell" window.

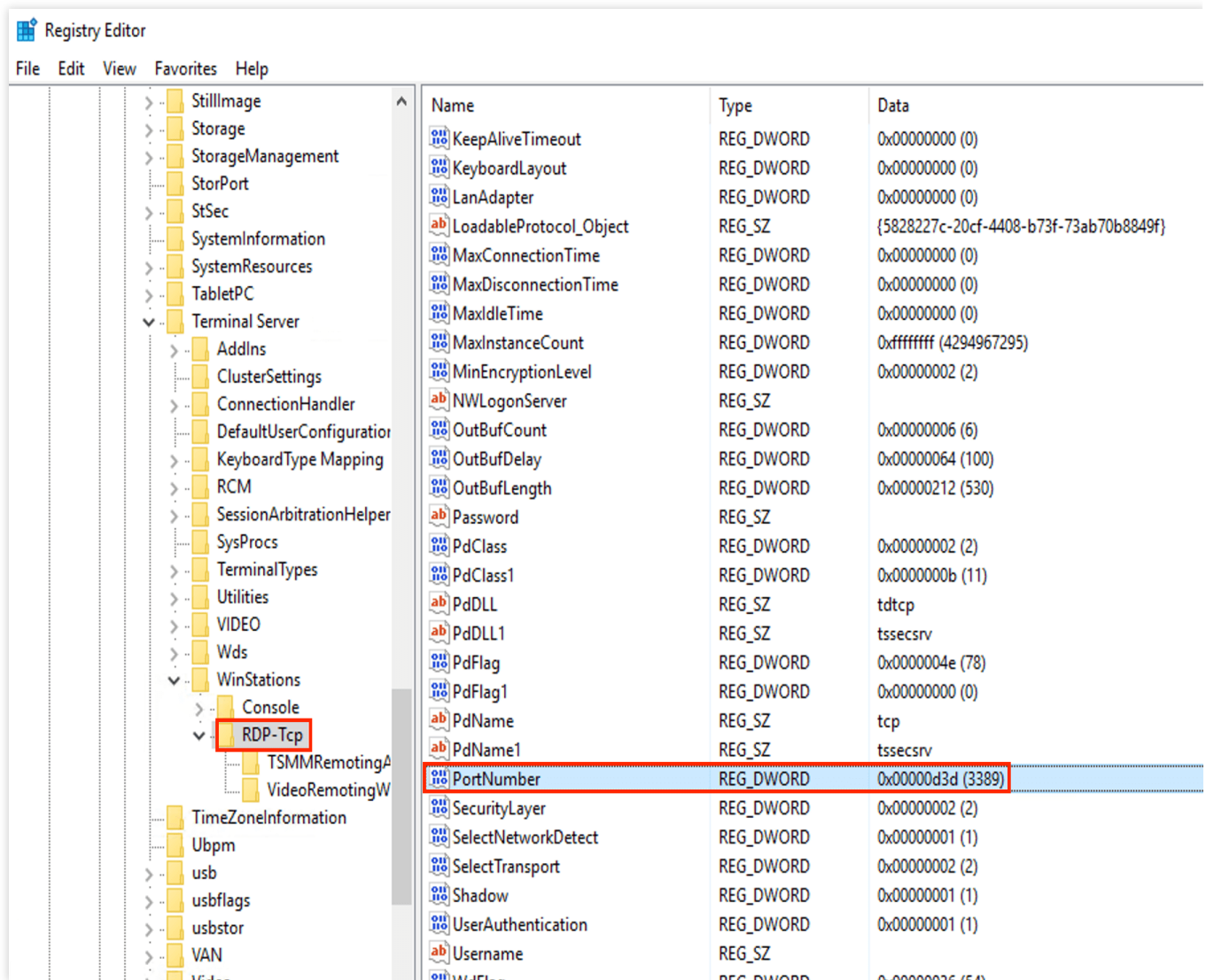
3. In the "Windows PowerShell" window, enter **regedit** and press **Enter** to open the "Registry Editor" window.
4. In the left-side registry navigation pane, expand the following hierarchies in sequence: **HKEY_LOCAL_MACHINE** > **SYSTEM** > **CurrentControlSet** > **Control** > **Terminal Server** > **Wds** > **rdpwd** > **Tds** > **tcp**.
- 5.

Find PortNumber in **tcp**

. Then, change the value of PortNumber from 3389 to an unoccupied port number within the range of 0 to 65535, as shown in the following figure.



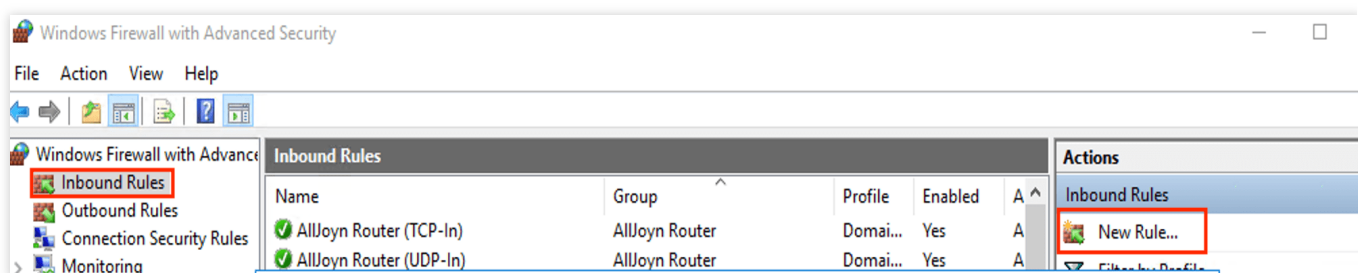
6. In the left-side registry navigation pane, expand the following hierarchies in sequence: **HKEY_LOCAL_MACHINE > SYSTEM > CurrentControlSet > Control > Terminal Server > WinStations > RDP-Tcp**.
7. Find and change PortNumber in **RDP-Tcp** to be the same as that in **tcp**.



8. (Optional) If a firewall is enabled for your CVM, be sure to add the new port to the allowlist of the firewall and set to allow connection.

8.1 In the “Windows PowerShell” window, enter **wf.msc** and press **Enter** to open the “Windows Firewall with Advanced Security” window.

8.2 In the “Windows Firewall with Advanced Security” window, select **Inbound Rules** and click **New rule**, as shown in the following figure.



8.3 On the “Rule Type” page in the “New Inbound Rule Wizard” window, select **Port** and click **Next**.

8.4 On the “Protocol and Ports” page in the **New Inbound Rule Wizard** window, select **TCP** and enter the port number set in [Step 5](#) in **Specific Port**. Then, click **Next**, as shown in the following figure.

The screenshot shows the 'New Inbound Rule Wizard' window with the 'Protocol and Ports' step selected in the left sidebar. The main area contains two questions with radio button options. The first question, 'Does this rule apply to TCP or UDP?', has 'TCP' selected. The second question, 'Does this rule apply to all local ports or specific local ports?', has 'Specific local ports:' selected, with a text box containing '12345'. Below the text box is an example: 'Example: 80, 443, 5000-5010'. At the bottom right are three buttons: '< Back', 'Next >' (highlighted with a blue border), and 'Cancel'.

New Inbound Rule Wizard

Protocol and Ports

Specify the protocols and ports to which this rule applies.

Steps:

- Rule Type
- Protocol and Ports**
- Action
- Profile
- Name

Does this rule apply to TCP or UDP?

☒ **TCP**

☐ **UDP**

Does this rule apply to all local ports or specific local ports?

☐ **All local ports**

☒ **Specific local ports:**

Example: 80, 443, 5000-5010

< Back **Next >** Cancel

8.5 On the **Action** page in the “New Inbound Rule Wizard” window, select **Allow connections** and click **Next**.

8.6 On the “Profile” page in the “New Inbound Rule Wizard” window, apply the default profile and click **Next**.

8.7 On the “Name” page in the “New Inbound Rule Wizard” window, enter the rule name and click **Finish**.

8.8 In the “Windows PowerShell” window, enter **services.msc** and press **Enter** to go to the “Services” window.

8.9 Find and right-click **Remote Desktop Services** in the “Services” window. Then, select **Restart** to restart the remote login service.

8.10 Refer to [Modifying Security Group Rules](#) to modify the security group rule with the protocol port “TCP:3389” by changing the port number to that set in [Step 5](#).

Inbound rule		Outbound rule				
Add a Rule Import rule Sort Delete Open all ports How to Set						
<input type="checkbox"/>	Source <small>?</small>	Protocol port <small>?</small>	Policy	Notes	Modify time <small>?</small>	Operation
<input type="checkbox"/>	0.0.0.0/0	TCP:3389	Allow	-	2020-05-12 18:15:09	Edit Insert C

Modifying the default remote port of a Linux CVM

Note:

Before you modify the default remote port, we recommend that you add the SSH port number and test whether the port is successfully connected to the CVM. Then, delete the default port 22. Ensure that the default port 22 cannot be connected to the CVM when the new port fails to connect to the CVM.

The following operations use CentOS 7.3 as an example. The specific operations vary slightly according to the version and language of the operating system.

1. [Log in to a Linux instance by using VNC.](#)
2. Run the following command to modify the configuration file:

```
vim /etc/ssh/sshd_config
```

3.

Press **i** to switch to the editing

mode and add a new port. Add `Port x` (where x is the port number of the new port) in a new row below `#Port 22`, and delete `#` to comment off `Port 22`, as shown in the following figure.

For example, add `Port 23456` in the row.

```
# If you want to change the port on a S
# SELinux about this change.
# semanage port -a -t ssh_port_t -p tcp
#
Port 22
Port 23456
#AddressFamily any
```

4. Press **Esc**, enter `:wq`, and save the change.
5. Run the following command so that the new configuration takes effect:

```
systemctl restart sshd.service
```

6. (Optional) Configure the firewall.

By default, the iptables service is used as the firewall for Linux CVMs with CentOS earlier than CentOS 7. Configure the firewall as follows if iptables rules have been configured for the CVM.

6.1.1 Run the following command to configure the firewall:

```
iptables -A INPUT -p tcp --dport <New port number> -j ACCEPT
```

For example, if the new port number is 23456, run the following command:

```
iptables -A INPUT -p tcp --dport 23456 -j ACCEPT
```

6.1.2 Run the following command to restart the firewall:

```
service iptables restart
```

The Firewalld service is used as the firewall for Linux CVMs with CentOS 7 or later. Configure the firewall as follows if the Firewalld service has been enabled on the CVM.

Run the following command to allow access by the port number added in [Step 3](#):

```
firewall-cmd --add-port=<New port number>/tcp --permanent
```

For example, if the new port number is 23456, run the following command:

```
firewall-cmd --add-port=23456/tcp --permanent
```

If `success` is returned, the port was successfully configured.

7. Refer to [Modifying Security Group Rules](#) to modify the security group rule with the protocol port “TCP:22” by changing the port number to that set in [Step 3](#).

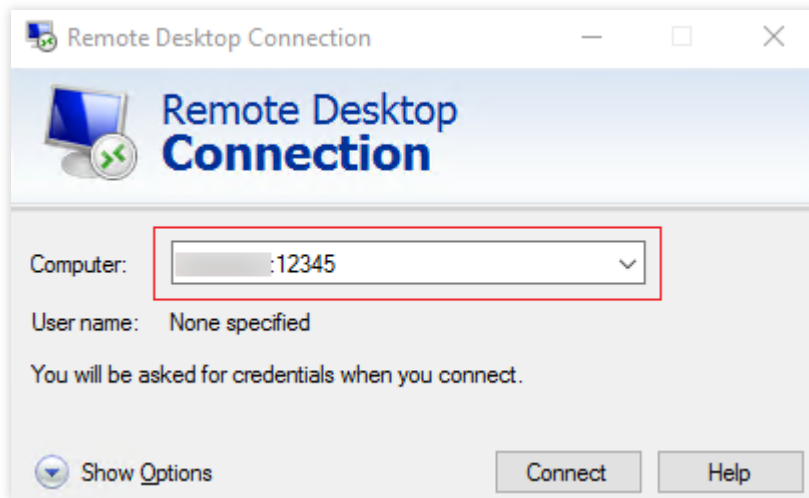
Inbound rule							Outbound rule
<div>Add a Rule Import rule Sort Delete Open all ports How to Set</div>							
<input type="checkbox"/>	Source i T	Protocol port i	Policy	Notes	Modify time i	Operation	
<input type="checkbox"/>	0.0.0.0/0	TCP:22	Allow	-	2020-05-12 18:15:09	Edit Insert C	

Verification

Windows CVMs

1. Assume that the Windows operating system is installed on the local computer. Open the “Remote Desktop Connection” dialog box.

2. Enter the `Internet IP of Windows server: port number after modification` after **Computer** and click **Connect**, as shown in the following figure.



3. Enter the admin account and password as prompted and click **OK**.

If the operating system interface of the Windows CVM appears, the connection is established.

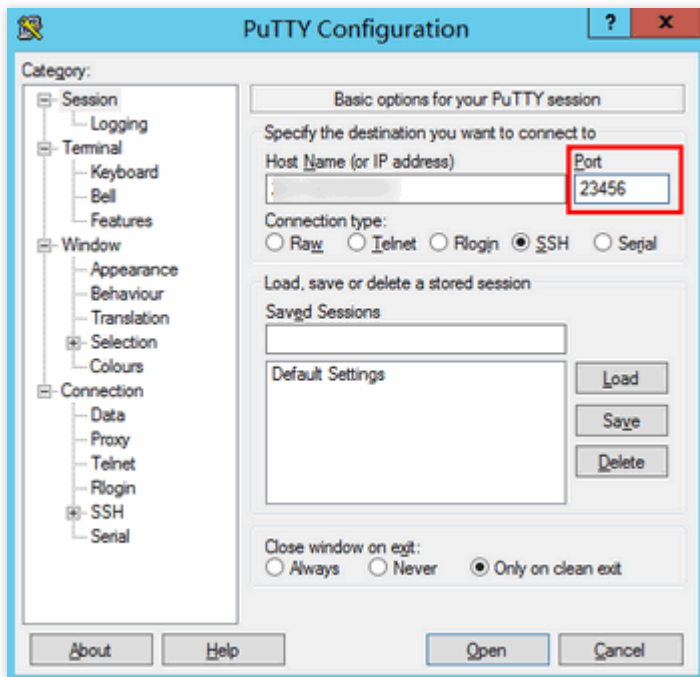
Note:

If you log in to the Windows CVM by using an RDP file, modify the `full address:s` parameter in the RDP file.

Linux CVMs

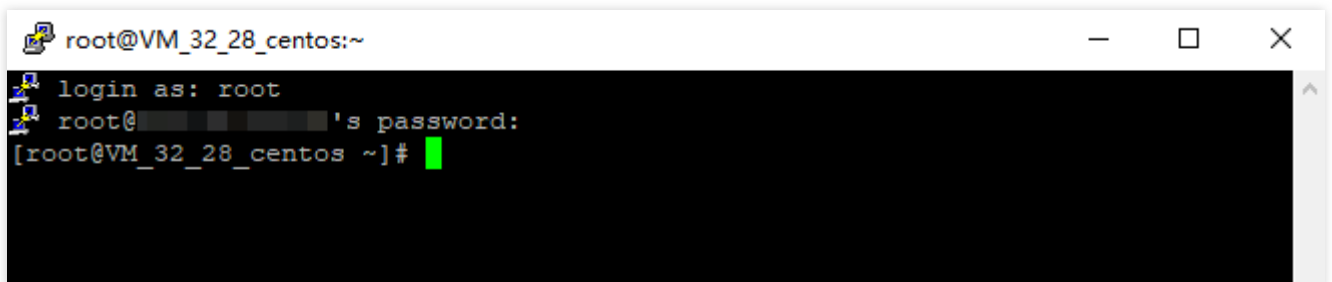
1. Assume that PuTTY is used for remote login. Start the PuTTY client.

2. In the "PuTTY Configuration" window, enter the public IP address of the Linux CVM, set **Port** to the new port number, and click **Open**, as shown in the following figure.



3. Enter the username and password of the Linux CVM as prompted and press **Enter**.

If the following output appears, the connection is established.



4. After using the new port to successfully establish a connection to the Linux CVM, run the following command:

```
vim /etc/ssh/sshd_config
```

5. Press **i** to switch to the editing mode, and add **#** in front of **Port 22** to comment off the port.

6. Press **Esc**, enter **:wq**, and save the change.

7. Run the following command so that the new configuration takes effect. Ensure that you use the new port for the next remote login to the Linux CVM.

```
systemctl restart sshd.service
```

Getting Instance Login Logs

Last updated : 2024-01-06 17:38:11

Overview

This document describes how to obtain user login records of a CVM instance.

Directions

Linux instances

Windows instances

Note:

This document uses a Linux instance on TencentOS Server 3.1 (TK4) as an example. As the steps may vary by operating system, proceed based on the actual conditions.

1. [Log in to the Linux instance by using Web Shell.](#)
2. You can view the following user login information as needed:

Note:

User login logs are usually stored in the `/var/run/utmp` , `/var/log/wtmp` , `/var/log/btmp` , and `/var/log/lastlog` files.

Run the `who` command to view the information of the currently logged-in user in the `/var/run/utmp` file. The returned result is as follows:

```
[root@VM-0-50-centos ~]# who
root      pts/0      2022-05-10 11:15 (81.49.102.31)
```

Run the `w` command to view the username of the currently logged-in user and display the currently executed tasks of the user in the `/var/run/utmp` file. The returned result is as follows:

```
[root@VM-0-50-centos ~]# w
11:36:28 up 39 min,  1 user,  load average: 0.00, 0.00, 0.00
USER      TTY      FROM          LOGIN@      IDLE        JCPU   PCPU WHAT
root      pts/0    81.49.102.31  11:15      0.00s      0.01s   0.00s w
```

Run the `users` command to view the username of the currently logged-in user in the `/var/run/utmp` file. The returned result is as follows:

```
[root@VM-0-50-centos ~]# users
root
```

Run the `last` command to view the information of the currently and historically logged-in users in the `/var/log/wtmp` file. The returned result is as follows:

```
[root@VM-0-50-centos ~]# last
root      pts/0      81.49.102.31    Tue May 10 11:15    still logged in
root      pts/0      81.49.102.29    Tue May 10 11:14 - 11:14    (00:00)

wtmp begins Tue May 10 11:14:11 2022
```

Run the `lastb` command to view the information of all the users who failed to log in to the system in the `/var/log/btmp` file. The returned result is as follows:

```
[root@VM-0-50-centos ~]# lastb
root      ssh:notty   104.35.203.49   Tue May 10 11:15 - 11:15    (00:00)

btmptmp begins Thu Oct 12 14:24:00 2034
```

Run the `lastlog` command to view the last user login information in the `/var/log/lastlog` file. The returned result is as follows:

```
[root@VM-0-50-centos ~]# lastlog
Username      Port      From      Latest
root          pts/0      81.49.102.31    Tue May 10 11:15:14 +0800 2022
bin                               **Never logged in**
daemon        **Never logged in**
adm           **Never logged in**
lp            **Never logged in**
sync          **Never logged in**
```

Run the `cat /var/log/secure` command to view the login information. The returned result is as follows:

```
[root@VM-0-69-centos ~]# cat /var/log/secure

May 10 14:25:36 VM-0-69-centos sshd[1075]: Received signal 15; terminating.
May 10 14:25:36 VM-0-69-centos sshd[2436]: Server listening on 0.0.0.0 port 22.
May 10 14:25:36 VM-0-69-centos sshd[2436]: Server listening on :: port 22.
May 10 14:25:42 VM-0-69-centos sshd[2484]: pam_unix(sshd:auth): authentication failure; logname= uid=0 euid=0 tty=ssh ruser= rhost=104.35.203.44 user=root
May 10 14:25:42 VM-0-69-centos sshd[2484]: pam_succeed_if(sshd:auth): requirement "uid >= 1000" not met by user "root"
May 10 14:25:44 VM-0-69-centos sshd[2484]: Failed password for root from 104.35.203.44 port 50226 ssh2
May 10 14:25:46 VM-0-69-centos sshd[2484]: Received disconnect from 104.35.203.44 port 50226:11: [preauth]
May 10 14:25:46 VM-0-69-centos sshd[2484]: Disconnected from authenticating user root 104.35.203.44 port 50226 [preauth]
May 10 14:25:51 VM-0-69-centos sshd[2519]: Accepted password for root from 85.49.100.35 port 45474 ssh2
May 10 14:25:51 VM-0-69-centos systemd[2541]: pam_unix(systemd-user:session): session opened for user root (uid=0)
May 10 14:25:51 VM-0-69-centos sshd[2519]: pam_unix(sshd:session): session opened for user root by (uid=0)
May 10 14:25:51 VM-0-69-centos sshd[2519]: pam_lastlog(sshd:session): corruption detected in /var/log/btmp
May 10 14:25:51 VM-0-69-centos sshd[2519]: error: lastlog_get_entry: Error reading from /var/log/lastlog: reading 292, got 1
```

Note:

This document uses a Windows instance on Windows Server 2012 R2 English as an example. As the steps may vary by operating system, proceed based on the actual conditions.

1. [Log in to the Windows instance by using WebRDP](#).

2. On the desktop, click



to open Server Manager.

3. In the **Server Manager** window, select **Tools > Event Viewer** in the top-right corner.

4. In the **Event Viewer** pop-up window, select **Windows Logs > Security** on the left and click **Filter Current Log** on the right.

5. In the **Filter Current Log** pop-up window, enter `4648` in <All Event IDs> and click **OK**.

6. In the **Event Viewer** window, double-click the logs that meet the filters.

7. In the **Event Property** pop-up window, click **Details** to view the client name and address as well as event recording time.