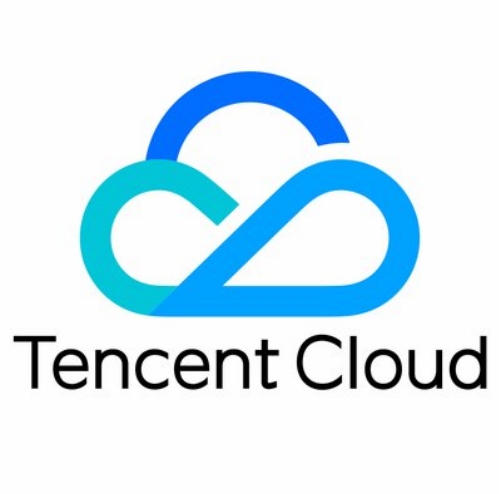
Cloud Virtual Machine

Guide for Operation and Maintenance

Product Introduction



Contents

Guide for Operation and Maintenance
Linux CVM Operation Manual
Common Operations and Commands

Last updated：2018-10-10 14:29:37

1. What is Linux server load average?

Load is used for measuring the workload of server, i.e. the length of the queue of tasks to be executed by CPU of the computer. The greater the value, the more processes that are currently running or waiting to be executed.
Reference: http://en.wikipedia.org/wiki/Load_average

2. How can I check Linux server load?

You can check it using w, top, uptime or procinfo commands, or from /proc/loadavg file.
Please refer to "Installing Software in Linux Environment" for the instructions on how to install procinfo tool.

3. What can I do when the server load is too high?

Server load/load average is displayed based on the length of process queue.
The high load of server (based on the average value over 15 minutes) may be caused by such reasons as insufficient CPU resources, I/O read/write bottleneck, insufficient memory resources, or the intensive computing that is being performed by CPU.
It is recommended to use vmstat -x, iostat and top commands to determine the reasons for the overload, and then find the processes that are taking up much of the resources for optimization.

4. How can I check server memory usage?

You can check this using free, top (press shift+m to sort by memory usage after the execution), vmstat, or procinfo commands, or from /proc/meminfo file.

5. How can I check the memory occupied by a single process?

You can check it using top -p PID, pmap -x PID, or ps aux|grep PID commands, or from /proc/$process_id (process' PID) /status file, e.g. /proc/7159/status file.

6. How can I check services and ports that are in use?

You can check this using netstat -tunlp, netstat -antup, or lsof -i:PORT commands.

7. How can I check server process information?

You can check this using ps auxww|grep PID, ps -ef, lsof v-p PID, or top -p PID commands.

8. How can I stop the process?

You can use kill -9 PID (process ID) and killall program name (e.g. killall cron) to kill a process.
If the process to be killed is a zombie process, then its parent process must be killed before it can be effectively killed. The command is: kill -9 ppid (ppid is parent process ID, which can be found with ps -o ppid PID, e.g. ps -o ppid 32535).

9. How can I find a zombie process?

You can use top command to check the total number of zombie processes, and use ps -ef | grep defunct | | grep -v grep to search for the information on specific zombie processes.

10. Why can't server port be enabled?

Enabling and listen-in of server port need to be checked from operating system itself and the application. Port below 1024 can only be enabled by root users on the Linux operating system. This means that you need to run sudo su first to obtain root permission before enabling the server port.
For any problem with application, it is recommended to use application startup log to identify reasons for failure, for example, port conflict (port used by Tencent server system cannot be occupied, such as 36000), configuration problem, etc.

11. What are the commands commonly used for checking the performance of a Linux server?

Command Name	Description
top	Process monitoring command for monitoring the overall performance of system. This command can be used to display information about system load, process, CPU, memory and paging. Shift+m and Shift+p are often used to sort processes by memory usage and CPU usage.
vmstat	System monitoring command which focuses on virtual memory but can also be used for monitoring the status information about CPU, process, memory paging and IO. For example, vmstat 3 10 outputs results every 3 seconds and is executed 10 times.
iostat	A tool used for outputting CPU and IO statuses and can display detailed IO information of system. For example, iostat -dxmt 10 outputs detailed information about IO in MB every 10 seconds.
df	Used to check the disk space usage of system. For example, df -m displays disk usage in MB.
lsof	List the files opened in the system. Since Linux is based on file system, this command is very useful in system management. For example: lsof -i: 36000 displays the processes using Port 36000 lsof -u root displays the programs run by root lsof -c php-fpm displays the files opened by php-fpm process lsof php.ini displays the processes for which php.ini is opened.
ps	A command for viewing process. It can be used to display details of the process. Commonly used command parameter combinations are ps -ef and ps aux. Ps -A -o is recommended to be used for customization of output fields. For example: ps -A -o pid,stat,uname,%cpu,%mem,rss,args,lstart,etime |sort -k6,6 -rn outputs results according to the listed fields and sorts by the 6th field ps -A -o comm |sort -k1 |uniq -c|sort -k1 -rn|head lists the process with the largest number of running instances.

Other often-used commands and files include free -m, du, uptime, w, /proc/stat, /proc/cpuinfo and /proc/meminfo.
Reference: http://en.wikipedia.org/wiki/Template:Unix_commands,http://www.linuxmanpages.com/

##12. What can I do when Cron does not work?
The trouble-shooting procedures are as follows:
1) Verify whether crontab is running normally.
You can run crontab -e command and add the following test entry */1 * * * * /bin/date >> /tmp/crontest 2>&1 &, and then observe /tmp/crontest file.
In case of any problem, it is recommended to use ps aux|grep cron to look for pid of cron and use kill -9 PID to terminate cron process, and then restart cron with /etc/init.d/cron start.

2) Verify whether the script path in the cron entry is an absolute path.

3) Check whether the user account for cron execution is correct, and check whether the account is included in /etc/cron.deny.

4) Check the execution permission of the script, script directory and log file permission.

5) It is recommended to run the script in background. Append a "&" to the script entry, for example, */1 * * * * /bin/date >> /tmp/crontest 2>&1 &

13. How can I set startup task for CVM?

Linux kernel startup procedure is as follows:
Start /sbin/init process,
then execute init initial script,
run level script/etc/rc.d/rc*.d, where value of * means running mode which can be viewed in /etc/inittab, and finally execute /etc/rc.d/rc.local.

The configuration of startup task can be made in S**rclocal file under /etc/rc.d/rc*.d, or in /etc/rc.d/rc.local.

14. Why is server disk read-only?

Common reasons for read-only hard disk are as follows:
1) Disk space is full
You can use df-m command to check the disk usage, and then delete unnecessary files to free disk space (Deletion of non-third party files is not recommended. Please verify it if required);
2) Disk inode resources are all occupied

You can use df-i command to check and verify related processes;
3) Hard disk failure

If hosting application is still unable to identify the reason using the above methods, please call the hot-line 4009100100 or submit ticket for assistance in locating.

15. How can I view Linux system logs?

The storage path for system-level log files is /var/log.
The commonly used system log is /var/log/messages.

16. How can I find large files in file system?

First, check disk partition usage with df commands, for example, df-m;
Then check the size of specific folder with du commands, for example, du -sh ./*, du -h --max-depth=1|head -10;
List files and their sizes using ls commands, for example, ls -lSh;
In addition, you can also directly check the size of files under specific directory by using find commands, for example, find / -type f -size +10M -exec ls -lrt {} \;

17. How can I check the version of server's operating system?

You can use the following command to check system version:
uname -a, cat /proc/version, cat /etc/issue

18. Why are there unreadable codes in the Chinese displayed by Linux terminal?

The server itself does not impose restrictions on the display language. If the display of Chinese is affected by the terminal software, you can try to adjust **Options** -> **Session Options** -> **Appearance** (secureCRT settings; please search for relevant settings for software of other versions.)
If the unreadable codes appear in pure Linux shell, please use export command to check settings for user environment variables and such environment variables as LANG and LC_CTYPE.

19. How can I set up the timeout for connection to CVM through SecureCRT?

It can be set as follows so that the connection is not broken when connecting to CVM through SecureCRT: Open secureCRT **Options**, choose **Session Options**, click **Terminal**, then check **Send protocol NO-OP** in the Anti-idle box on the right, and set the time to **every 120 seconds**.

20. Why isn't disk space freed after a file on Linux server is deleted?

Sometimes, after logging in to Linux server and executing rm command to delete a file on it, you may find that available disk space does not increase when you execute df command to check disk space. This is because when the file is deleted with rm command, if another process happens to be assessing the file, the space occupied by the deleted file will not be immediately freed at the time you check the disk space using df command.

Solution:
Use root permission to execute lsof |grep deleted, view the PID of the process which is using the deleted file, and then kill the process with kill -9 PID command.

Commands for First Launching

Last updated：2018-08-06 15:41:53

When **launching a CVM instance for the first time**, you can pass user data to the CVM by passing a text (with no format restriction) and execute the text.
This document uses Linux CVM as an example to describe how to output "Hello Tencent Cloud" by passing a Shell script when launching the CVM for the first time.
The log file (`/var/log/cloud-init-output.log`) output by Cloud-init catches the output of the console.

Notes

- A command can be executed by passing a text only on the first time of launching a CVM.
- The passed text must be encoded with Base64. **Please encode in Linux environment to avoid format incompatibility.
- The text, which is input as the user data, is executed using the root permission. Therefore, sudo command is not required in the script. Note: All the files you created belong to root. If you need to grant non-root users with the access permission, modify the corresponding permission in the script.
- Adding these tasks to the startup of the CVM will increase its startup time. Wait a few minutes until the tasks complete, and then test whether they are executed successfully.
- In this example, Shell script must start with `#!` and the path directing to the interpreter of the script to be read.

Step 1: Write Shell script

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

Note:
Shell script must start with `#!` and the path directing to the interpreter of the script to be read. For more information on Shell script, please see BASH Programming of the Linux Documentation Project (tldp.org).

Step 2: Encode the script file with Base64

> **Note:**
> Please encode in Linux environment to avoid format incompatibility.

Suppose that the script file you created in step 1 is script_text. You can encode the file using Base64 command in Linux environment, as shown below:

```
# Encode the file with Base64
base64 script_text

# The encoded result :
IyEvYmluL2Jhc2gKCmVjaG8gIldlbGNvbWUgVG8gVGVuY2VudCBDbG91ZC4iCg==

# Decode the returned result with Base64 and verify whether it is the command to be executed.
echo "IyEvYmluL2Jhc2gKZWNobyAiSGVsbG8gVGVuY2VudCBDbG91ZC4iCg==" | base64 -d
```

Step 3: Pass the text

We provide multiple methods to launch an instance and here we introduce two of them. Please choose a method as needed:

Passing on the official website or the console

When you create a CVM on the official website or the console, select **Advanced Configuration** in **4. Set Security Group and CVM** step. Enter the encoded result (`IyEvYmluL2Jhc2gKCmVjaG8gIldlbGNvbWUgVG8gVGVuY2VudCBDbG91ZC4iCg==` in this example) of step 2 in user defined data item. Finish the creation and launch the CVM.
Tencent Cloud CVM executes the script using the open-source software cloud-init. For more information

on cloud-init, please see cloud-init's official website.

Passing via API

When creating a CVM via API, you can pass the text by assigning the value of the encoded result of step 2 to UserData parameter of RunInstances API. The following is an example of the parameter of CVM creation request with UserData.

```
https://cvm.tencentcloudapi.com/?Action=RunInstances
&Version=2017-03-12
&Placement.Zone=ap-guangzhou-2
&ImageId=img-pmqg1cw7
&UserData=IyEvYmluL2Jhc2gKCmVjaG8gIldlbGNvbWUgVG8gVGVuY2VudCBDbG91ZC4iCg==
&<Common request parameters>
```

Mount Data Disks
Format and Mount Data Disks

Last updated：2018-08-06 15:39:45

Note:

- After formatting, all the data in the data disk will be cleared. Before formatting, make sure there is no data in the data disk or important data has been backed up. To avoid service exceptions, ensure that the CVM has stopped external services before formatting.

Manual Formatting and Mounting Data Disk

Format and mount a data disk by following the steps below.

Note:

- When executing the following commands, remember to modify the data disk drive letter. The drive letter is `vdb` in this example.

Step 1: Format the data disk

Note:
When formatting partitions, developers can decide the file system format on their own, such as `ext3` and `ext4`. `ext4` is used in this example.

Format the data disk by performing the mkfs command:

```
mkfs.ext4 /dev/vdb
```

Step 2: Mount the data disk

1. Create a mount point - data directory:

```
mkdir /data
```

2. Mount new partitions:

```
mount /dev/vdb /data
```

3. Verify whether the data disk is mounted successfully:

```
df -h
```

The following message indicates that it is mounted successfully (i.e. the data disk is mounted on the Linux CVM):

```
Filesystem Size Used Avail Use% Mounted on
/dev/vdb 50G 53M 47G 1% /data
```

Step 3: Enable auto mount upon launch

Add the data disk mount information to `/etc/fstab` to enable auto mount upon launch.

To allow your CVM to be automatically mounted with data disk when it is restarted or launched, add the mount information to `/etc/fstab` . Otherwise, the data disk cannot be automatically mounted to the CVM when the CVM is restarted or launched.

1. Execute the following command to add partition information:

```
echo '/dev/vdb /data ext4 defaults 0 0' >> /etc/fstab
```

2. Execute the following command to view the partition information:

```
cat /etc/fstab
```

The following message indicates that the data disk mount information is added successfully.

```
/dev/vdb /data ext4 defaults 0 0
```

Auto Formatting and Mounting Data Disk

You can format and mount a data disk on Tencent Cloud Linux CVM by running the following Shell script:

```
#!/bin/bash
type=ext4
mount_dir=/data
mkfs.$type /dev/vdb
mkdir -p $mount_dir
echo "/dev/vdb $mount_dir $type defaults 0 0" >> /etc/fstab
mount -a
```

Read/write NTFS Data Disks after Reinstalling a Windows CVM to Linux CVM

Last updated：2018-07-17 17:22:31

Windows file system typically uses NTFS or FAT32 format, while Linux file system often uses EXT series format. When the operating system is reinstalled and changed from Windows to Linux, its type has changed but the data disk remains the old format. Thus, denied access to the data disk file system may occur in the reinstalled system. You can perform the following operations on the reinstalled Linux CVM to read data from the data disk of the original Windows system:

1) Use the following command to install ntfsprogs software on the Linux system so that Linxu can support NTFS file system:

```
yum install ntfsprogs
```

2) Mount the data disk under Windows to Linux CVM. Skip this step if the data disk has already been mounted.

Log in to Tencent Cloud console, enter "Cloud Virtual Machine" - "Cloud Block Storage" tab, click on the Windows data disk to be mounted, and then click "More" - "Mount to Cloud Virtual Machine" button. Select reinstalled Linux CVM in the pop-up box, then click "Confirm".

3) Use `parted -l` command to check the data disk mounted from Windows:

```
Model: Virtio Block Device (virtblk)
Disk /dev/vde: 21.5GB
Sector size (logical/physical): 512B/512B
Partition Table: gpt

Number  Start   End     Size    File system  Name                          Flags
 1      17.4kB  134MB   134MB                Microsoft reserved partition  msftres
 2      135MB   3331MB  3196MB  ntfs         Basic data partition
```

4) Use `mount -t ntfs-3g data disk path mount point' command to mount the data disk:

```
[root@VM_127_193_centos ~]# mount -t ntfs-3g /dev/vde2 mnt/
[root@VM_127_193_centos ~]# ls mnt/
$RECYCLE.BIN  test.txt
```

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

Environment Configurations
LNMP Environment Configurations for SUSE

Last updated：2017-11-10 09:45:21

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:

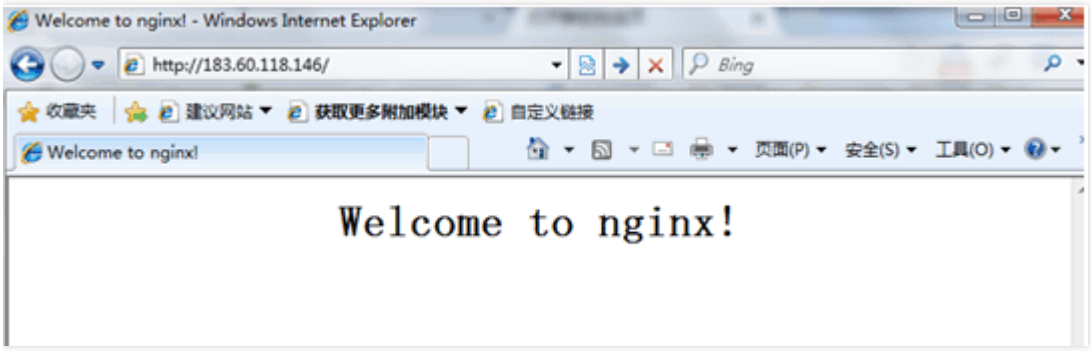
```
wget http://127.0.0.1
```

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:

```
VM_137_55_sles10_64:~ # /etc/init.d/mysql start; /etc/init.d/php-fpm start; /etc/init.d/nginx start
Starting MySQL                                                done
Starting php-fpm                                              done
Starting nginx Checking for service nginx                     running
                                                              done
```

4. Environment configuration validation

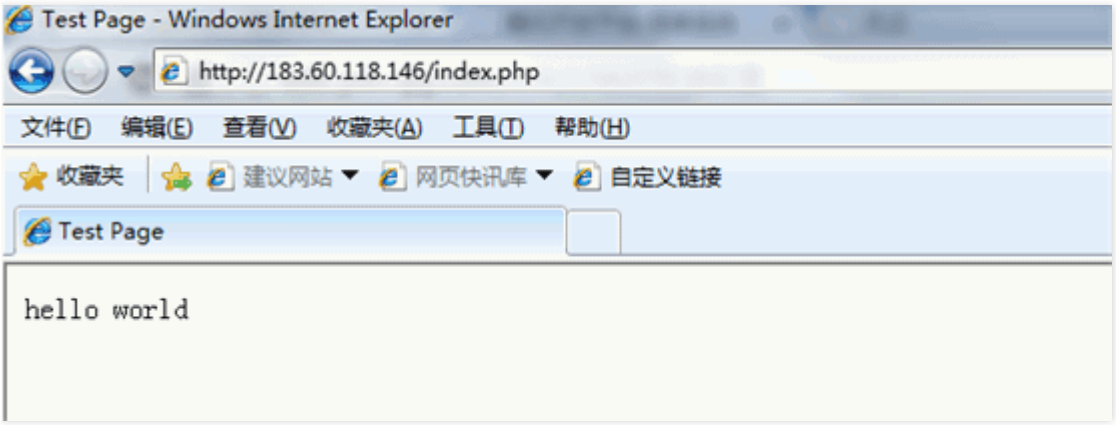
Create index.php under a web directory using the following command:

```
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.



Linux Power Management Configuration

Last updated：2017-09-19 22:30:21

A Linux system without an acpi management program will suffer failures of soft shutdown. Therefore, make sure that the acpi (power management for Linux) module has been installed on your CVM.

Checking method

Check whether the acpi has been installed using the following command:

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

If there's no such process, it hasn't been installed. Then you need to follow the next step to install the module. If there's such process, the next step can be ignored.

Installation method

Use the following command to install the acpi module.

1) For Ubuntu/Debian system

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

2) For Redhat/CentOS system

```
yum install acpid
```

3) For SUSE system

```
in apcid
```

Note: The CoreOS system doesn't have such problem.

Reset Passwords of Activated Linux CVMs

Last updated：2018-06-25 11:44:47

If you need to reset password for a batch of Linux CVMs without shutting them down, you can download the reset script (Click here to download) to batch reset password online.

> Note: If you run the script on a machine of public network, the ip added to the hosts.txt file must be the Public IP of the host. If the script is run on the private network CVM of Tencent Cloud, you can fill in the Private IP of the host.

The using method of script is as follows.

Input the ip of CVM to be operate on, ssh port, account, old and new passwords into the hosts.txt file. Each line represents a host, for example:

```
10.0.0.1 22 root old_passwd new_passwd
10.0.0.2 22 root old_passwd new_passwd
```

Run the following code:

```
./batch-chpasswd.py
```

Example of returned results:

```
------------------------------------------------------------
change password for root@10.0.0.1
spawn ssh root@10.0.0.1 -p 22
root's password:
Authentication successful.
Last login: Tue Nov 17 20:22:25 2015 from 10.181.225.39
[root@VM_18_18_centos ~]# echo root:root | chpasswd
[root@VM_18_18_centos ~]# exit
logout
------------------------------------------------------------
change password for root@10.0.0.2
spawn ssh root@10.0.0.2 -p 22
root's password:
Authentication successful.
Last login: Mon Nov 9 15:19:22 2015 from 10.181.225.39
[root@VM_19_150_centos ~]# echo root:root | chpasswd
```

```
[root@VM_19_150_centos ~]# exit
logout
```

Set up FTP Service

Last updated：2018-08-06 15:40:53

This document describes how to build FTP service on Linux CVM. In this example, CentOS 7.2 64-bit system is used for illustration. vsftpd is used as FTP server and FileZilla as the client.

Step 1: Install vsftpd

1. Log in to the CVM.

2. Install the software. Enter the command:

```
yum install vsftpd -y
```

3. If "Complete !" is displayed, the installation is completed.

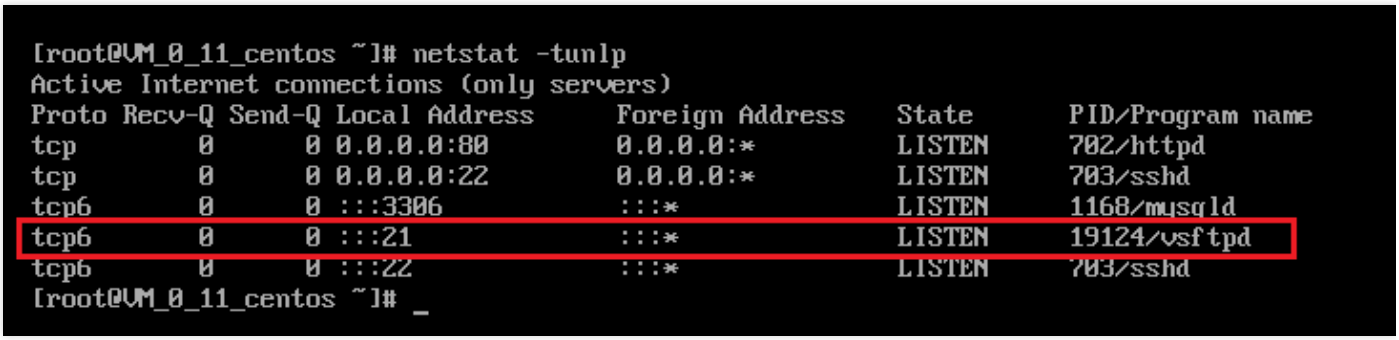
Step 2: Start vsftpd service

1. Start the service. Enter the command:

```
systemctl start vsftpd
```

2. Confirm whether it is started with a command. Enter the following command. If the result is displayed as shown below, the service has been started.

```
netstat -tunlp
```



3. Confirm whether it is started through public network access.

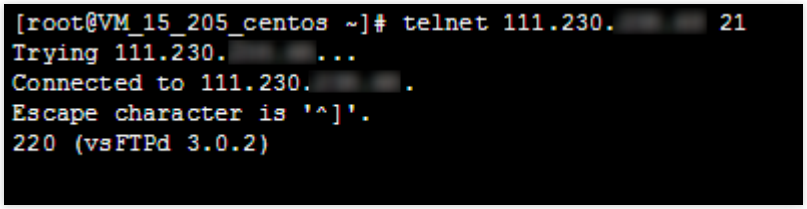
   i. Install telnet service with the following command.

   ```
   yum -y install telnet
   ```

   ii. Test with a command on another computer connected to the Internet:

   ```
   telnet + CVM public network IP + 21
   ```

   If the following is displayed, the service has been started.



Step 3: Edit vsftpd configuration file

1. In CVM, enter the command: `vi /etc/vsftpd/vsftpd.conf`

2. Edit the content. Change the status to "anonymous login is not allowed". Press "a" on the keyboard to start editing. Change `anonymous_enable=YES` in the file to `anonymous_enable=NO`. After the modification, press "Esc" on the keyboard, and enter `:write` anywhere to save the changes. Enter `:quit`

to quit the editing.

```
#
# READ THIS: This example file is NOT an exhaustive list of vsftpd options.
# Please read the vsftpd.conf.5 manual page to get a full idea of vsftpd's
# capabilities.
#
# Allow anonymous FTP? (Beware - allowed by default if you comment this out).
anonymous_enable=NO
#
# Uncomment this to allow local users to log in.
# When SELinux is enforcing check for SE bool ftp_home_dir
local_enable=YES
```

Step 4: Add FTP users

1. Add users. In this example, a user named ftpuser1 is added. Enter the command: `useradd -m -d /home/ftpuser1 -s /sbin/nologin ftpuser1`

2. Set user login password. In this example, login password is set for user ftpuser1. Enter the command: `passwd ftpuser1` . Enter the password and confirm it.

```
[root@VM_0_11_centos ~]# useradd -m -d /home/ftpuser1 -s /sbin/nologin ftpuser1
[root@VM_0_11_centos ~]# passwd ftpuser1
Changing password for user ftpuser1.
New password:
Retype new password:
Sorry, passwords do not match.
New password:
Retype new password:
passwd: all authentication tokens updated successfully.
```

FAQ

Problem description

Some users may encounter such problem as connection timeout and failure to read the directory list when using FTP client connections locally, as shown below.

The problem occurs at the PASV command. The reason is that FTP protocol is incompatible with Tencent Cloud network architecture. FTP client transmits data in passive mode by default. Therefore, it searches for the server's IP address to connect during the communication process. However, public IP of Tencent Cloud is not directly configured on ENI, so the client cannot find a valid IP in passive mode (it can only find private IP of CVM. The private IP cannot communicate directly with the public network). Therefore, the connection cannot be established.

Solution

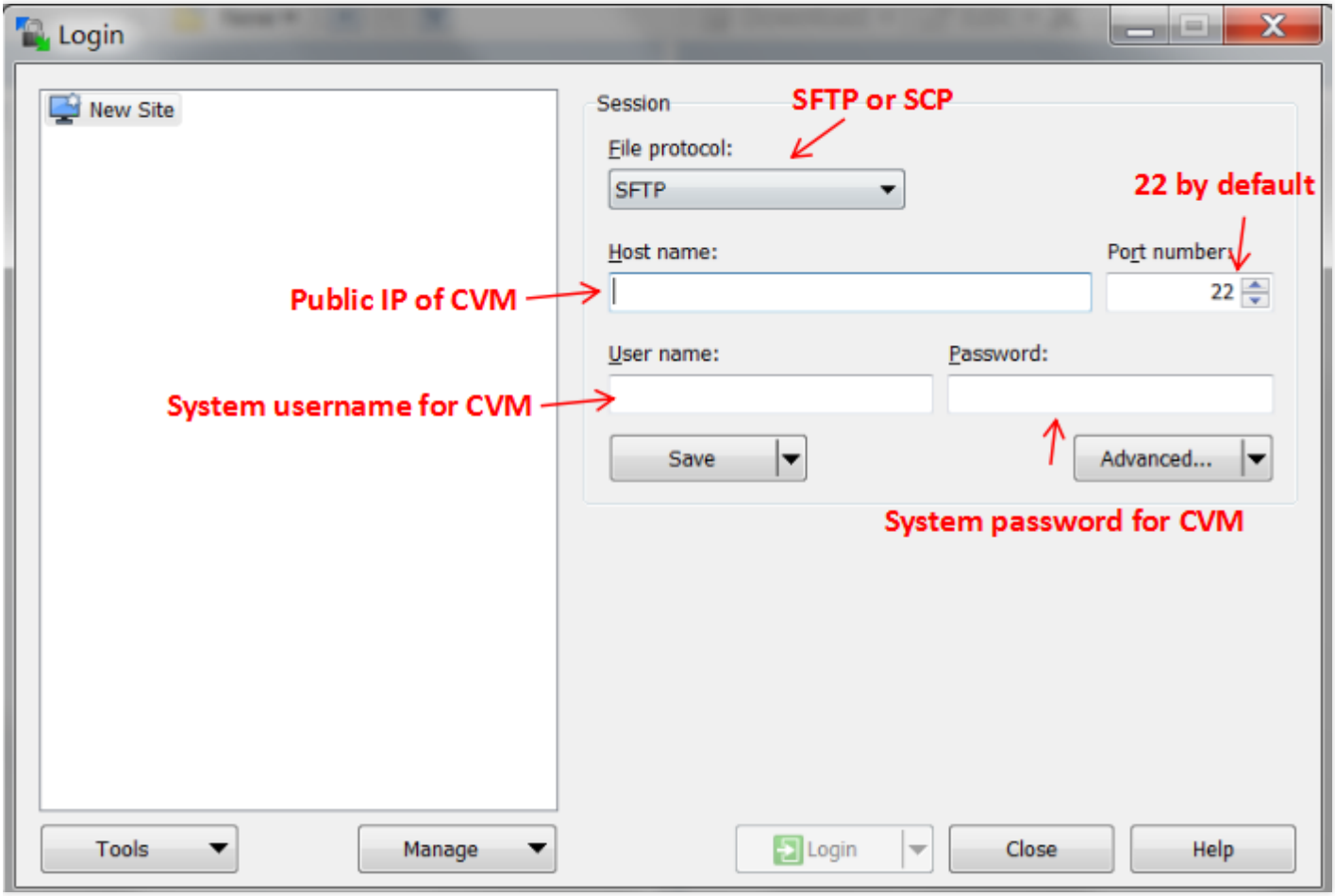
- Change the transmission mode on the client to active.
- If the client network environment requires passive mode, then add the following statements to the configuration file of the server Step 3:

```
pasv_address=XXX.XXX.XXX.XXX // (Public IP)
pasv_enable=YES
pasv_min_port=1024
pasv_max_port=2048
```

Code Deployment
Upload Files via WinSCP

Last updated：2018-04-28 10:46:54

WinSCP is an open source graphical SFTP client that uses SSH in Windows environment and supports SCP protocol. Its main function is to copy files between the local and remote computers safely. Instead of using FTP to upload code, you can use the server account and password to access the server directly via WinSCP, without any configuration on the server side. Download address: Official Download.
Start WinSCP after installation. The interface is as follows. Fill in the information as shown and log in.



How to fill in the fields:

- Protocol: either SFTP or SCP is OK
- Host Name: Public IP of CVM (Log into CVM Console to view the Public IP of CVM)

- Username: the system username for CVM (SUSE/CentOS/Debian: root, Windows: Administrator, Ubuntu: ubuntu)
- Password: the password corresponding to the username of CVM
- Port: 22 by default

Click on Log In after completing the information. After successful login, select a local file and drag it to the remote site on the right, and then you can upload the file to the Linux CVM.

Upload Files via FTP

Last updated：2017-11-09 11:33:19

You can use FTP channel to upload application from your own server to CVM.

1. Configure FTP service on CVM

1) Run the following commands as root to install Vsftp (take CentOS system as an example):

```
yum install vsftpd
```

2) Before starting the vsftpd service, you need to log into the CVM to modify configuration files to disable anonymous login.

Open the configuration file with the following command:

```
vim /etc/vsftpd/vsftpd.conf
```

Change

```
anonymous_enable=YES (on the 11th line in the configuration file)
```

to

```
anonymous_enable=NO
```

to disable anonymous login.

3) Read the effective configuration.

```
cat /etc/vsftpd/vsftpd.conf |grep ^[^#]
```

The following results will be returned:

```
local_enable=YES
write_enable=YES
local_umask=022
anon_upload_enable=YES
anon_mkdir_write_enable=YES
anon_umask=022
dirmessage_enable=YES
```

```
xferlog_enable=YES
connect_from_port_20=YES
xferlog_std_format=YES
listen=YES
pam_service_name=vsftpd
userlist_enable=YES
tcp_wrappers=YES
```

4) Start vsftpd service.

```
service vsftpd start
```

5) Set up an FTP user account.

Set up an FTP user account by running the following command:

```
useradd
```

For example, if the account is "ftpuser1", the directory is /home/ftpuser1, and login via ssh is not allowed:

```
useradd -m -d /home/ftpuser1 -s /sbin/nologin ftpuser1
```

And set a password for the account using the following command:

```
passwd
```

For example, setting the password for the above account as "ftpuser1":

```
passwd ftpuser1
```

After setting these up, you can log on to the FTP server using the account.

6) Modify the pam configuration of vsftpd, so that users can connect to the CVM via the account and password they set by themselves.

Use the following command to modify the pam:

```
vim /etc/pam.d/vsftpd
```

Modify to:

```
#%PAM-1.0
auth required /lib64/security/pam_listfile.so item=user sense=deny file=/etc/ftpusers onerr=succeed
auth required /lib64/security/pam_unix.so shadow nullok
auth required /lib64/security/pam_shells.so
```

```
account required /lib64/security/pam_unix.so
session required /lib64/security/pam_unix.so
```

Confirm whether the modified file is correct using the following command:

```
cat /etc/pam.d/vsftpd
```

Returned results are:

```
auth required /lib64/security/pam_listfile.so item=user sense=deny file=/etc/ftpusers onerr=succeed
auth required /lib64/security/pam_unix.so shadow nullok
auth required /lib64/security/pam_shells.so
account required /lib64/security/pam_unix.so
session required /lib64/security/pam_unix.so
```

Restart the vsftpd service using the following command to make the modification effective:

```
service vsftpd restart
```

The results are:

```
Shutting down vsftpd: [ OK ]
Starting vsftpd for vsftpd: [ OK ]
```

2. Upload files to Linux CVM

1) Download and install open source software FileZilla
Please use FileZilla Ver. 3.5.1 or 3.5.2 (Using FileZilla Ver. 3.5.3 for FTP uploading will lead to problems). Since FileZilla official site only provides the latest Ver.3.5.3 for download, you are recommended to search for download links for Ver.3.5.1 or 3.5.2 on your own. Recommended download link for Ver. 3.5.1: http://www.oldapps.com/filezilla.php?old_filezilla=6350

2) Connect to FTP
Run FileZilla, fill in setting form, and then click "Quick Links".

Description of the settings:

- Host: Public network IP of CVM (Log in to CVM Console page to view the public network IP of CVM).

- User Name: ID of the FTP user account set in the previous step (here “ftpuser1” is used as example).
- Password: Password of the FTP user account set in the previous step (here “ftpuser1” is used as example).
- Port: FTP listener port, default is "21".

3) Upload files to Linux CCVM

When uploading a file, select the local file with the mouse and drag it to the remote site to upload it to Linux CVM.

Note: CVM FTP path does not support automatic unzipping or deletion of uploaded tar zip files.

Upload Files via SCP

Last updated：2017-11-13 09:56:10

Linux machine can upload files to Linux CVM with the following commands:

```
scp local file address CVM login name@CVM public network IP/domain name CVM file location
```

For example, upload local file "/home/lnmp0.4.tar.gz" to the directory for the CentOS CVM with IP of 129.20.0.2:

```
scp /home/lnmp0.4.tar.gz root@129.20.0.2 /home/lnmp0.4.tar.gz
```

Press "Enter" and type in login password to complete the upload.

Installing Software
Tencent Cloud Software Source Acceleration Package Download and Update

Last updated：2018-08-13 11:36:50

To solve the problem that the access to the official source is too slow during software dependencies installation, Tencent Cloud has built cache service for some software. You can accelerate the installation of dependency packages using Tencent Cloud software origin server, and CVM without public network egress can use a software origin server via a private network to facilitate construction of service architecture. Tencent Cloud software origin server supports public network access and private network access.

Public network domain name

```
http://mirrors.cloud.tencent.com/
```

Private network domain name

```
http://mirrors.tencentyun.com/
```

The following are example illustrations based on private network domain names. If you access these software sources via a public network, replace the private network domain name with a public network domain name.

Accelerating pip Using Tencent Cloud Image Source

Temporary Use

Before **use**, make sure you have installed python.

Execute the following command to use Tencent Cloud pypi software source:

```
pip install -i http://mirrors.tencentyun.com/pypi/simple <some-package>
```

Note: `simple` in the path must be added.

Set as Default

Modify the file `~/.pip/pip.conf` (create one if it does not exist). Update `index-url` to Tencent Cloud path, for example:

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

Synchronization Period

Tencent Cloud makes synchronization from `pypi.python.org` official website every day.

Accelerating Maven Using Tencent Cloud Image Source

Before **use**, make sure you have installed JDK **and** Maven

Setting Method

Open Maven setting file `settings.xml` and configure the following repository mirror:

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

Accelerating npm Using Tencent Cloud Image Source

Before **use**, make sure you have installed Node.js **and** npm.

Setting Method

Execute the following command:

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

Accelerating docker Using Tencent Cloud Image Source

Tencent Cloud Container Service (CCS) Cluster

When creating a node, the CVM in CCS cluster automatically installs docker service and configure Tencent Cloud private network image, without the need of manual configuration.

Tencent Cloud CVM

> Make sure you have installed docker on CVM. Only Docker 1.3.2 **and** above support Docker Hub Mirr or mechanism. If you have **not** installed Docker **or** the installed version **is** too low, please install it **or** upgrade your version.

- For such systems as Ubuntu 14.04, Debian, CentOS 6, Fedora, OpenSUSE. The configuration may be slightly different in other versions:
  Modify Docker configuration file `/etc/default/docker`

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

- For Centos 7:
  Modify Docker configuration file `/etc/sysconfig/docker`

```
OPTIONS='--registry-mirror=https://mirror.ccs.tencentyun.com'
```

- For Windows:
  When using Boot2Docker, enter Boot2Docker Start Shell and execute:

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

  Restart Boot2Docker

Accelerating MariaDB Using Tencent Cloud Image

1. Configure MariaDB yum repo file
   Create `MariaDB.repo` file under `/etc/yum.repos.d/` (CentOS 7 is taken as an example and the actual address of operating system yum repos prevails):

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

   Write the following:

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

2. Execute command `yum clean all`
3. Install MariaDB with yum
   Execute `yum install MariaDB-client MariaDB-server`

Accelerating MongoDB Using Tencent Cloud Image

CentOS and Redhat Systems

In the example, MongoDB V3.4 is installed. If you need to **install** another **version**, **change** the **version** number **in** the mirror path.

1. Create file `/etc/yum.repos.d/mongodb.repo` and write the following content:

```
[mongodb-org-3.4]
name=MongoDB Repository
baseurl=http://mirrors.tencentyun.com/mongodb/yum/redhat/$releasever/3.4/
gpgcheck=0
enabled=1
```

2. Install mongodb

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

Debian System

In the example, MongoDB V3.4 is installed. If you need to **install** another **version**, **change** the **version** number **in** the mirror path.

1. Import MongoDB GPG public key

```
sudo apt-key adv --keyserver hkp://keyserver.ubuntu.com:80 --recv 0C49F3730359A14518585931BC711F9BA15703C6
```

2. Configure mirror path

```
#Debian7
echo "deb http://mirrors.tencentyun.com/mongodb/apt/debian wheezy/mongodb-org/3.4 main" | sudo tee /etc/apt/sources.list.d/mongodb-org-3.4.list
#Debian8
echo "deb http://mirrors.tencentyun.com/mongodb/apt/debian jessie/mongodb-org/3.4 main" | sudo tee /etc/apt/sources.list.d/mongodb-org-3.4.list
```

3. Install mongodb

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

Ubuntu System

In the example, MongoDB V3.4 is installed. If you need to **install** another **version**, **change** the **version** number **in** the mirror path.

1. Import MongoDB GPG public key

```
sudo apt-key adv --keyserver hkp://keyserver.ubuntu.com:80 --recv 0C49F3730359A14518585931BC711F9BA15703C6
```

2. Configure mirror path

```
#Ubuntu 12.04
echo "deb [ arch=amd64 ] http://mirrors.tencentyun.com/mongodb/apt/ubuntu precise/mongodb-org/3.4 multiverse" | sudo tee /etc/apt/sources.list.d/mongodb-org-3.4.list
#Ubuntu 14.04
```

```
echo "deb [ arch=amd64 ] http://mirrors.tencentyun.com/mongodb/apt/ubuntu trusty/mongodb-org/3.4 multiverse" | sudo tee /etc/apt/sources.list.d/mongodb-org-3.4.list
#Ubuntu 16.04
echo "deb [ arch=amd64,arm64 ] http://mirrors.tencentyun.com/mongodb/apt/ubuntu xenial/mongodb-org/3.4 multiverse" | sudo tee /etc/apt/sources.list.d/mongodb-org-3.4.list
```

3. Install mongodb

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

Accelerating Rubygems Using Tencent Cloud Image Source

`Make sure that you have installed Ruby locally`

Modifying Configuration

Execute the following command to modify RubyGems source address.

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

Synchronization Period

Tencent Cloud makes synchronization from `https://rubygems.org/` official website every day.

Install Software via Apt-get under Ubuntu Environment

Last updated：2018-03-15 19:41:38

To enhance users' software installation efficiency on CVM and reduce the costs for downloading and installing software, Tencent Cloud provides you with Apt-get download source. Users of CVM on the operating system of Ubuntu12.04 can quickly install software through Apt-get.

For apt-get download source, software package can be installed directly without adding software source. In order to speed up software installation, the system has already configured mirror of Ubuntu for private network. The mirror is a full image of official x86_64 and is in line with the source of official website.

1. Installation steps

1) Log into the CVM on the operating system of Ubuntu12.04

2) Use the following command to install the software:

```
sudo apt-get install
```

Examples are as follows:

```
sudo apt-get install nginx php5-cli php5-cgi php5-fpm php5-mcrypt php5-mysql mysql-client-core-5.5 mysql-server-core-5.5
```

Result:

```
root@VM-144-105-ubuntu:~# sudo apt-get install nginx php5-cli php5-cgi php5-fpm php5-mcrypt php5-mysql mysql-c
p5-mcrypt php5-mysql mysql-client-core-5.5 mysql-server-core-5.5
Reading package lists... Done
Building dependency tree
Reading state information... Done
The following extra packages will be installed:
  libmcrypt4 libmysqlclient18 mysql-common nginx-common nginx-light php5-common
Suggested packages:
  libmcrypt-dev mcrypt php-pear php5-suhosin
The following NEW packages will be installed:
  libmcrypt4 libmysqlclient18 mysql-client-core-5.5 mysql-common
  mysql-server-core-5.5 nginx nginx-common nginx-light php5-cgi php5-cli
  php5-common php5-fpm php5-mcrypt php5-mysql
0 upgraded, 14 newly installed, 0 to remove and 62 not upgraded.
Need to get 22.1 MB of archives.
After this operation, 66.3 MB of additional disk space will be used.
Do you want to continue [Y/n]?
```

3) Input "Y" to confirm and start the installation until the software is installed.

2. View the information of the installed software

After the software has been installed, you can view the installation directory of the software package and all the files within the package using the following command:

```
sudo dpkg -L
```

The following command can be used to view the version information of the software package:

```
sudo dpkg -l
```

Examples are as follows:

```
sudo dpkg -L nginx
sudo dpkg -l nginx
```

The results are as follows (The actual version may be different from this one; please refer to the version actually queried):

```
root@VM-144-105-ubuntu: # sudo dpkg -L nginx
/.
/usr
/usr/share
/usr/share/doc
/usr/share/doc/nginx
/usr/share/doc/nginx/README.Debian
/usr/share/doc/nginx/copyright
/usr/share/doc/nginx/changelog.Debian.gz
/usr/share/doc/nginx/CHANGES.gz
```

```
root@VM-139-150-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                 Description
+++-=====================-=======================-================================================================
ii  nginx                 1.1.19-1ubuntu0.2       small, but very powerful and efficient web server and mail pro
```

Install Software via YUM under CentOS Environment

Last updated：2018-10-08 10:00:40

To enhance users' software installation efficiency on CVM and reduce the costs for downloading and installing software, Tencent Cloud provides you with Yum download source. In CentOS environment, users can quickly install software through YUM.

For Yum download source, software package can be installed directly without adding software source.

1. Installation steps

1) After logging into the CVM on a CentOS operating system, root permission is granted by default:

> Note: The execution of password command is forbidden, and the root password cannot be modified by default.

2) Run the following command as root to install the software:

```
yum install [nginx][php][php-fpm][mariadb][mariadb-server][mysql][mysql-server]...
```

> Note: Since CentOS 7, MariaDB has become the default database installer in yum source. MySQL will be unusable when installed in a system higher than CentOS 7 via Yum. You can choose to use MariaDB that is fully compatible, or refer to here to install MySQL of a lower version.

3) System will automatically search relevant software packages and dependencies, and prompt users to verify whether the software package is suitable in the interface as shown below:

```
================================================================================================================
 Package                          Arch                Version                              Repository
================================================================================================================
Installing:
 mariadb                          x86_64              1:5.5.44-2.el7.centos                os
 mariadb-server                   x86_64              1:5.5.44-2.el7.centos                os
Installing for dependencies:
 perl-Compress-Raw-Bzip2          x86_64              2.061-3.el7                          os
 perl-Compress-Raw-Zlib           x86_64              1:2.061-4.el7                        os
 perl-DBD-MySQL                   x86_64              4.023-5.el7                          os
 perl-DBI                         x86_64              1.627-4.el7                          os
 perl-Data-Dumper                 x86_64              2.145-3.el7                          os
 perl-IO-Compress                 noarch              2.061-2.el7                          os
 perl-Net-Daemon                  noarch              0.48-5.el7                           os
 perl-PlRPC                       noarch              0.2020-14.el7                        os
Updating for dependencies:
 mariadb-libs                     x86_64              1:5.5.44-2.el7.centos                os

Transaction Summary
================================================================================================================
Install  2 Packages (+8 Dependent packages)
Upgrade             ( 1 Dependent package)

Total download size: 22 M
Is this ok [y/d/N]: y_
```

4) Input "y" to confirm and start the installation. When the installation is done, it will display "Complete" as shown below:

```
  Installed:
    mariadb.x86_64 1:5.5.44-2.el7.centos                    mariadb-server.x86_64 1:5.5.44-2.el7.centos

  Dependency Installed:
    perl-Compress-Raw-Bzip2.x86_64 0:2.061-3.el7                perl-Compress-Raw-Zlib.x86_64 1:2.061-4.el7
    perl-DBD-MySQL.x86_64 0:4.023-5.el7                         perl-DBI.x86_64 0:1.627-4.el7
    perl-Data-Dumper.x86_64 0:2.145-3.el7                       perl-IO-Compress.noarch 0:2.061-2.el7
    perl-Net-Daemon.noarch 0:0.48-5.el7                         perl-PlRPC.noarch 0:0.2020-14.el7

  Dependency Updated:
    mariadb-libs.x86_64 1:5.5.44-2.el7.centos

  Complete!
```

2. View the information of the installed software

After the software has been installed, you can view the installation directory of the software package using the following command:

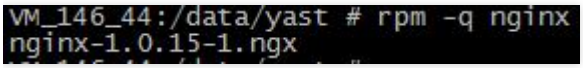
```
rpm -ql
```

Taking the installation directory of nginx as an example:

The following command can be used to view the version information of the software package:

```
rpm -q
```

Taking the version of nginx as an example (The actual version may be different from this one; please refer to the version actually queried):



Install Software via zypper under SUSE Environment

Last updated：2017-11-29 22:52:18

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:

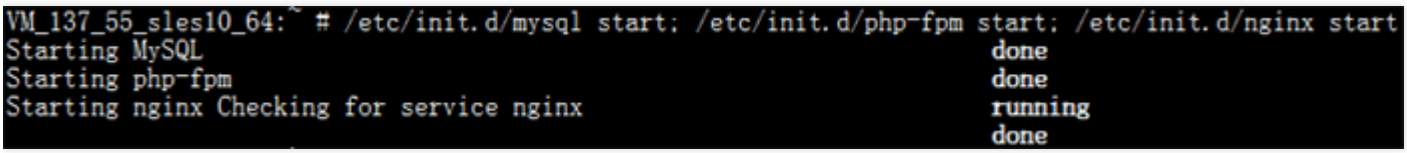
```
[global]
error_log = /var/log/php-fpm.log
[www]
user = nobody
group = nobody
listen = 127.0.0.1:9000
pm = dynamic
pm.max_children = 5
pm.start_servers = 2
pm.min_spare_servers = 1
pm.max_spare_servers = 3
```

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:



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.

Access Internet
Allow CVMs withtout Internet access to access Internet

Last updated：2017-10-19 21:52:30

When the CVM chooses 0Mbps bandwidth, the public network cannot be accessed. The CVM can only access the external network through a CVM with a Public IP.

1. Principle

- A CVM without a Public IP can access the public network through a CVM with a Public IP by using proxy on a CVM with a Public IP or via vpn.
- The proxy is easy to configure but complicated to use. It is suggested that you use pptp vpn to do this. (i.e., A CVM without a Public IP can be connected with a CVM with a Public IP through pptp protocol, and the CVM with a Public IP will be set to the gateway in pptp network)

2. Configuration

Assume that a CVM with a Public IP is A, and a CVM without a Public IP is B.

1) Install pptpd on A, on CentOS for example (other Linux release versions are similar) using the following command:

```
yum install pptpd
```

2) Modify the configuration file /etc/pptpd.conf by adding the following two lines

```
localip 192.168.0.1
remoteip 192.168.0.234-238,192.168.0.245
```

3) Modify the configuration file /etc/ppp/chap-secrets by adding the username and password (the 1st column indicates the username, and the 3rd column indicates the password)

```
user pptpd pass *
```

4) Start services

```
service pptpd start
```

5) Enable the forward capability

```
# echo 1 > /proc/sys/net/ipv4/ip_forward
# iptables -t nat -A POSTROUTING -o eth0 -s 192.168.0.0/24 -j MASQUERADE
```

6) Install the client on B, on CentOS for example, using the following command:

```
# yum install pptp pptp-setup
```

7) Create a configuration file

```
# pptpsetup --create pptp --server 10.10.10.10 --username user --password pass --encrypt
```

Note: --server is followed by A's IP address.

8) Connect pptpd

```
# pppd call pptp
```

9) Set the route:

```
# route add -net 10.0.0.0/8 dev eth0
# route add -net 172.16.0.0/12 dev eth0
# route add -net 192.168.0.0/16 dev eth0
# route add -net 0.0.0.0 dev ppp0
```

In addition, if B is Windows CVM, a network "Connecting to Workspace" can be created to connect to the pptpd server

Windows CVM Operation Manual Commands for First Launching

Last updated：2018-08-06 15:45:48

When launching a CVM instance for the first time, you can pass user data to the CVM by passing a text and execute the text.
This document uses Windows CVM as an example to describe how to output "Hello Tencent Cloud" by passing a PowerShell script when launching the CVM for the first time.

Notes

- A command can be executed by passing a text only on the first time of launching a CVM.
- The passed text must be encoded with Base64.
- Adding these tasks to the startup of the CVM will increase its startup time. Wait a few minutes until the tasks complete, and then test whether they are executed successfully.
- In this example, specify the Windows PowerShell script using PowerShell label. For example:

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

Step 1: Write PowerShell script

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

Step 2: Encode the script file with Base64

```
$Bytes = [System.Text.Encoding]::Unicode.GetBytes($Script)
$EncodedText =[Convert]::ToBase64String($Bytes)

# The encoded result:
PABwAG8AdwBlAHIAcwBoAGUAbABsAD4AIAAiAEgAZQBsAGwAbwAgAFQAZQBuAGMAZQBuAHQAI
```

```
ABDAGwAbwB1AGQALgAiACAAfAAgAE8AdQB0AC0ARgBpAGwAZQAgAC4AXAB0AGUAbgBjAGUAbg
B0AGMAbABvAHUAZAAuAHQAeAB0ACAAPAAvAHAAbwB3AGUAcgBzAGgAZQBsAGwAPgA=
```

Step 3: Pass the text

Passing on the official website or the console

When you create a CVM on the official website or the console, select **Advanced Configuration** in **4. Set Security Group and CVM** step. Enter the encoded result of the step 2 in user defined data item. Finish the creation and launch the CVM.
For example, the encoded result in this example is as follows:

```
PABwAG8AdwBlAHIAcwBoAGUAbABsAD4AIAAiAEgAZQBsAGwAbwAgAFQAZQBuAGMAZQBuAHQAI
ABDAGwAbwB1AGQALgAiACAAfAAgAE8AdQB0AC0ARgBpAGwAZQAgAC4AXAB0AGUAbgBjAGUAbg
B0AGMAbABvAHUAZAAuAHQAeAB0ACAAPAAvAHAAbwB3AGUAcgBzAGgAZQBsAGwAPgA=
```

Passing via API

When creating a CVM via API, you can pass the text by assigning the value of the encoded result of step 2 to UserData parameter of RunInstances API. The following is an example of the parameter of CVM creation request with UserData.

```
https://cvm.tencentcloudapi.com/?Action=RunInstances
&Version=2017-03-12
&Placement.Zone=ap-guangzhou-2
&ImageId=img-pmqg1cw7
&UserData=PABwAG8AdwBlAHIAcwBoAGUAbABsAD4AIAAiAEgAZQBsAGwAbwAgAFQAZQBuAGMA
ZQBuAHQAIABDAGwAbwB1AGQALgAiACAAfAAgAE8AdQB0AC0ARgBpAGwAZQAgAC4AXAB0AGUA
bgBjAGUAbgB0AGMAbABvAHUAZAAuAHQAeAB0ACAAPAAvAHAAbwB3AGUAcgBzAGgAZQBsAGwA
PgA=
&<Common request parameters>
```

Format Data Disks of Windows CVMs Data Disk Partition and Formating of Windows CVMs

Last updated：2017-10-25 15:07:24

By default, the data disks purchased on the CVM purchase page are not automatically mounted under an offline state. Data disks that are not partitioned and formatted cannot be used. This tutorial will guide you to mount, partition and format data disks in a Windows system.

The path to the "Disk Management" interface may vary with the Windows version (Windows 2012, Windows 2008, Windows 2003, etc.), but the steps to partition and format the disks are basically the same.

This article provides the guide on how to mount, partition and format data disks on Windows 2012 and Windows 2008.

Note:

Once formatted, all the data in the disk will be cleared. Make sure that there is no data left in the disk or the important data has been backed up before formatting. To avoid any service exception, make sure that the CVM has stopped providing services before formatting.

1. Disk Partitioning and Formatting on Windows 2012

On Windows 2012, the path to Disk Management is "Start" - "Server Management" - "Tools" - "Computer Management" - "Disk Management".

"Disk 1" is an unpartitioned disk. Here, the process is illustrated by creating one partition for "Disk 1". Right click on Disk 1, then select "Online". Right click again, then select "Initialize Disk". Select "GPT" or "MBR" depending on the partitioning method, and click on the "OK" button.

Note: Make sure to select GPT as the partitioning method if the disk is larger than 2TB.

Right click on the unallocated space, and select "New Simple Volume". In the "New Simple Volume Wizard" pop-up window, click "Next".

Enter the desired disk size for the partition, then click "Next". Enter the drive letter, then click "Next". Select "File System", then "Format Partition", and click "Next". Upon completing the New Simple Volume Wizard, click "Finish".

2. Disk Partitioning and Formatting on Windows 2008

On Windows 2008, the path to "Disk Management", different from that on Windows 2012, is "Server Management" - "Storage" - "Disk Management".

"Disk 1" is an unallocated disk. Here, the process is illustrated by creating one partition for "Disk 1".

"Disk 1" is not online in the initial state. Right click "Disk 1", and then click "Online" in the pop-up menu.

Again, right click "Disk 1", and then click "Initialize Disk" in the pop-up menu.

Select the GPT initialization method, and click the "OK" button.

Note: Make sure to select GPT as the partitioning method if the disk is larger than 2TB.

Right click on the unallocated region behind "Disk 1", and select "New Simple Volume" in the shortcut menu that pops up.
As prompted by the Wizard, enter the size of the disk partition, then click "Next".

Select "File System", then "Format Partition", and click "Next".

Upon completing the New Simple Volume Wizard, click "Finish".

"Formatting..." is displayed.

At this point, the newly partitioned data disk can be seen on the computer screen.

> Note: Do not convert a basic hard disk to a dynamic hard disk. We are not liable for any data loss arising out of this action.

3. Online Settings

Under a Windows operating system, online settings are often needed in Disk Management. To help you make better use of Elastic Cloud Block Storage, we recommend that you modify the operating system as follows:
Open the cmd line and run the following command

```
diskpart
san policy = onlineall
```

Once remounted to the Windows CVM, the Elastic Cloud Block Storage can be used directly without any user action as long as it contains a valid file system.

Read/write EXT Data Disks after Reinstalling a Linux CVM to Windows CVM

Last updated：2017-12-15 15:32:34

The file system format of Windows is generally NTFS or FAT32, while that of Linux is EXT series. When the operating system is reinstalled and changed from Linux to Windows, its type has changed but the data disk remains the old format. Thus, denied access to the data disk file system may occur in the reinstalled system. You can perform the following operations on the reinstalled Windows CVM to read data from the data disk of the original Linux system:

1) Assume that the data disk of Linux CVM has two partitions:

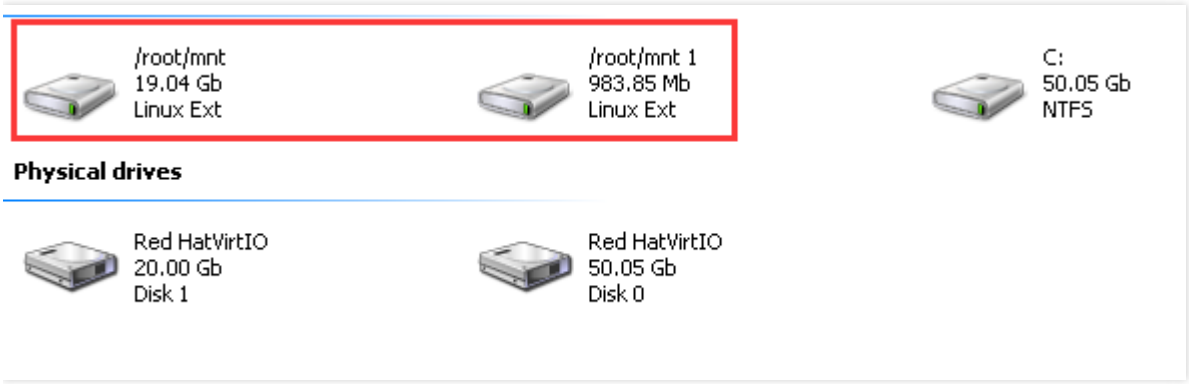
```
Disk /dev/vdb: 21.5 GB, 21474836480 bytes
16 heads, 63 sectors/track, 41610 cylinders
Units = cylinders of 1008 * 512 = 516096 bytes
Sector size (logical/physical): 512 bytes / 512 bytes
I/O size (minimum/optimal): 512 bytes / 512 bytes
Disk identifier: 0x29cc8ca2

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

2) Download and install DiskInternals Linux Reader software on the reinstalled Windows CVM (For download address, please click here).

3) Mount the data disk under Linux to Windows CVM. Skip this step if the data disk has already been mounted.

Log into Tencent Cloud console, enter "Cloud Virtual Machine" - "Cloud Block Storage" tab, click on the Linux data disk, and then click "More" - "Mount to Cloud Virtual Machine" button. Select reinstalled Windows CVM in the pop-up box, then click "Confirm".

4) Click to run DiskInternals, and you can see the information of data disk just mounted. /root/mnt and /root/mnt1 are for partitions vdb1 and vdb2 respectively:



5) Click to enter /root/mnt, and right-click the file you want to copy, and select Save to save the file.

6) Please note that the Linux data disk is read-only at this time. If you need to perform read and write operations on the data disk as Windows data disk, please first back up the files you need and then re-format it into a standard type supported by Windows operating system. For specific operations, please see here.

Installing Software
Install and Configure IIS

Last updated：2017-11-09 11:29:41

NOTE: Do not install any anti-virus software of PC type on Windows CVM. Such software may block the telnet port of the CVM, making it impossible to log in to the CVM.

1. Installation and configuration of IIS

1.1. Example for Windows2012R2

1) Click "Start" at the bottom left corner of Windows CVM, select "Server Manager" to open the Server Manager interface.

2) Select "Add Roles and Features", then in "Before You Begin" in the "Add Roles and Features Wizard" pop-up box, click "Next". In "Installation Type", select "Role-based or Feature-based Installation", then click "Next".

3) In the left side of the window, select "Server Role" tab, check "Web Server (IIS)", click "Add Features" button in the pop-up box, and then click "Next".

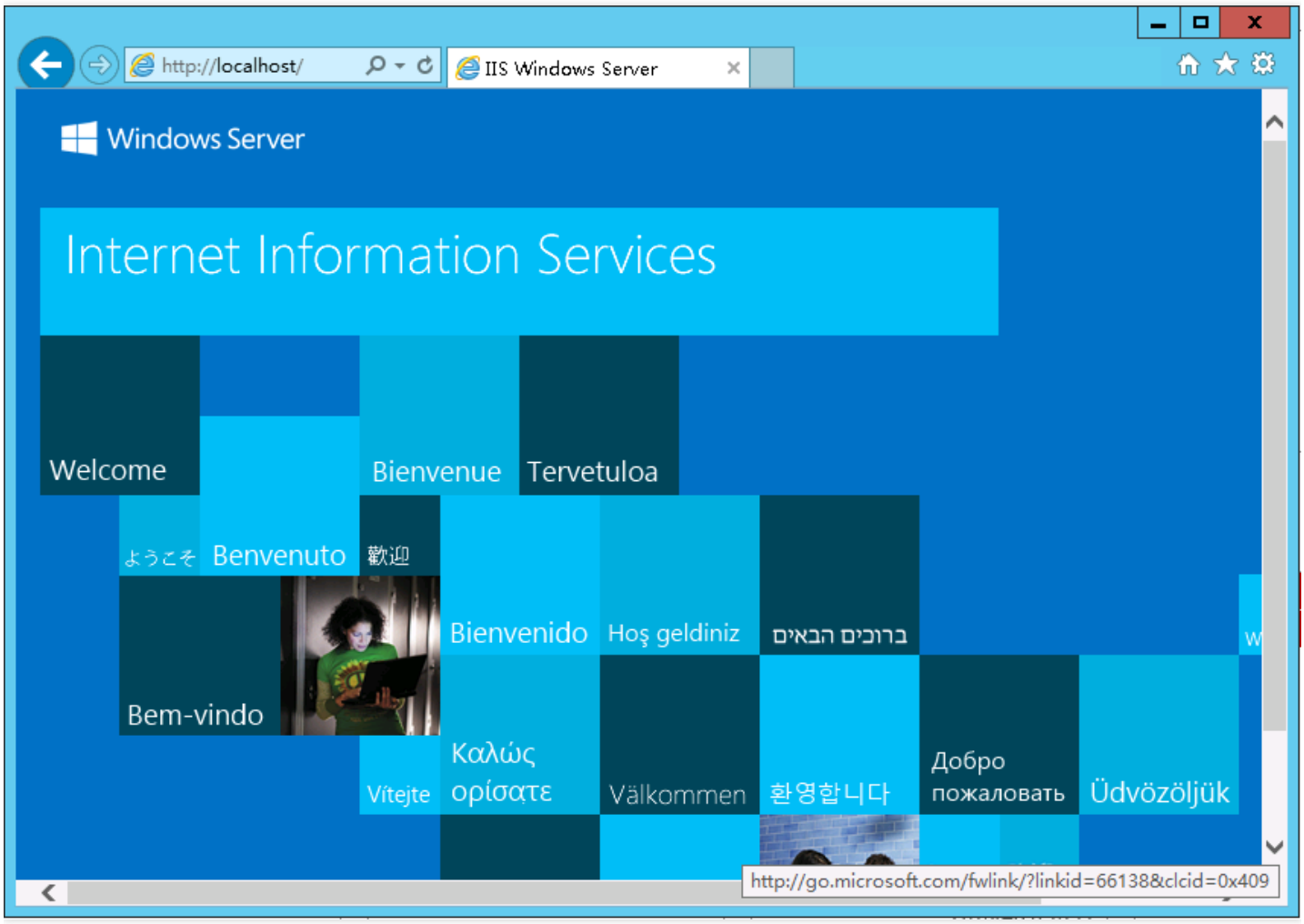
4) In the "Features" tab, click "Next", and in the "Web Server Role (IIS)" tab, also click "Next".

5) In the "Role Services" tab, check the "CGI" option, then click "Next".

6) Confirm the installation and wait for the installation to be completed.

7) When the installation has been completed, access localhost in the browser of CVM to verify whether the installation is successful. The appearance of the following page indicates that the installation has

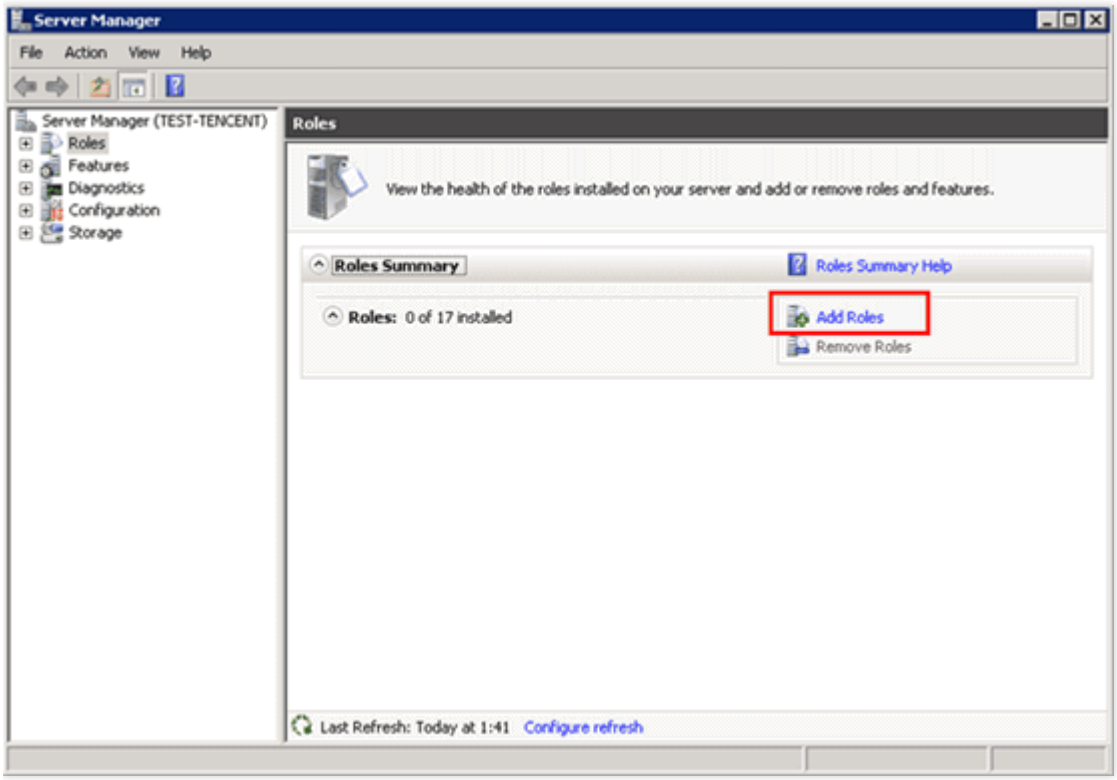
been completed successfully.

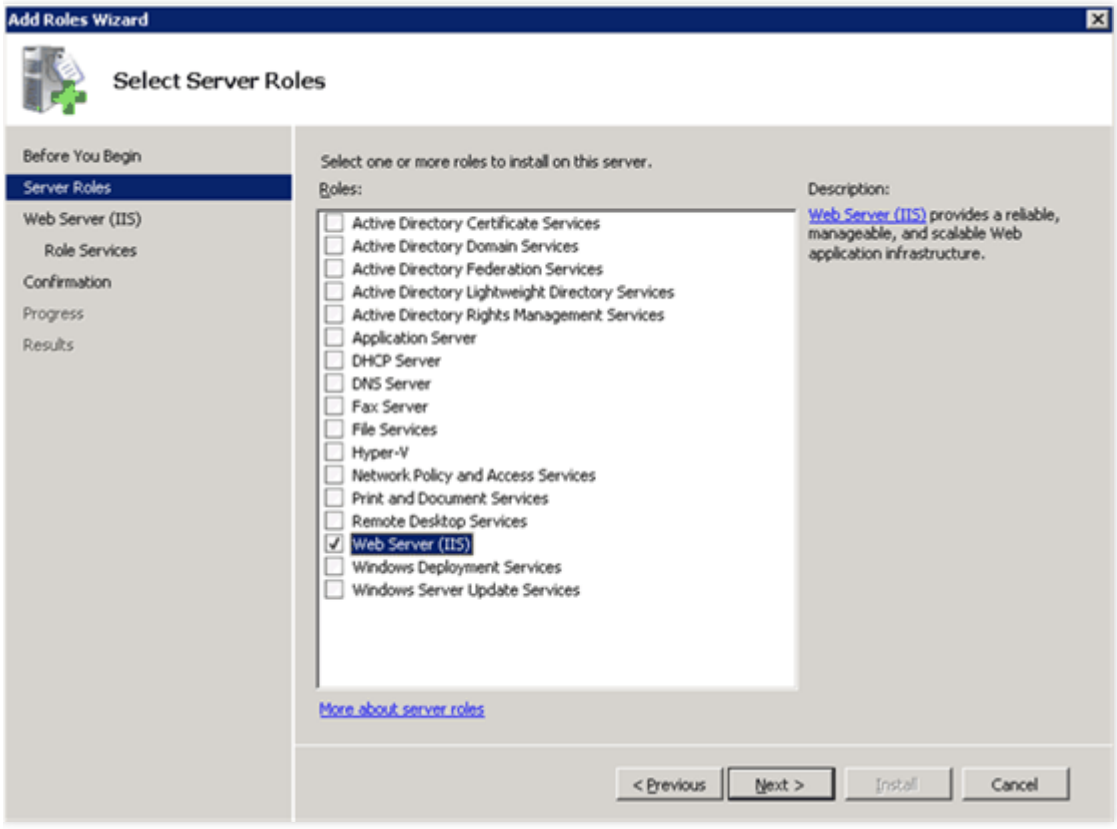


1.2. Example for Windows2008

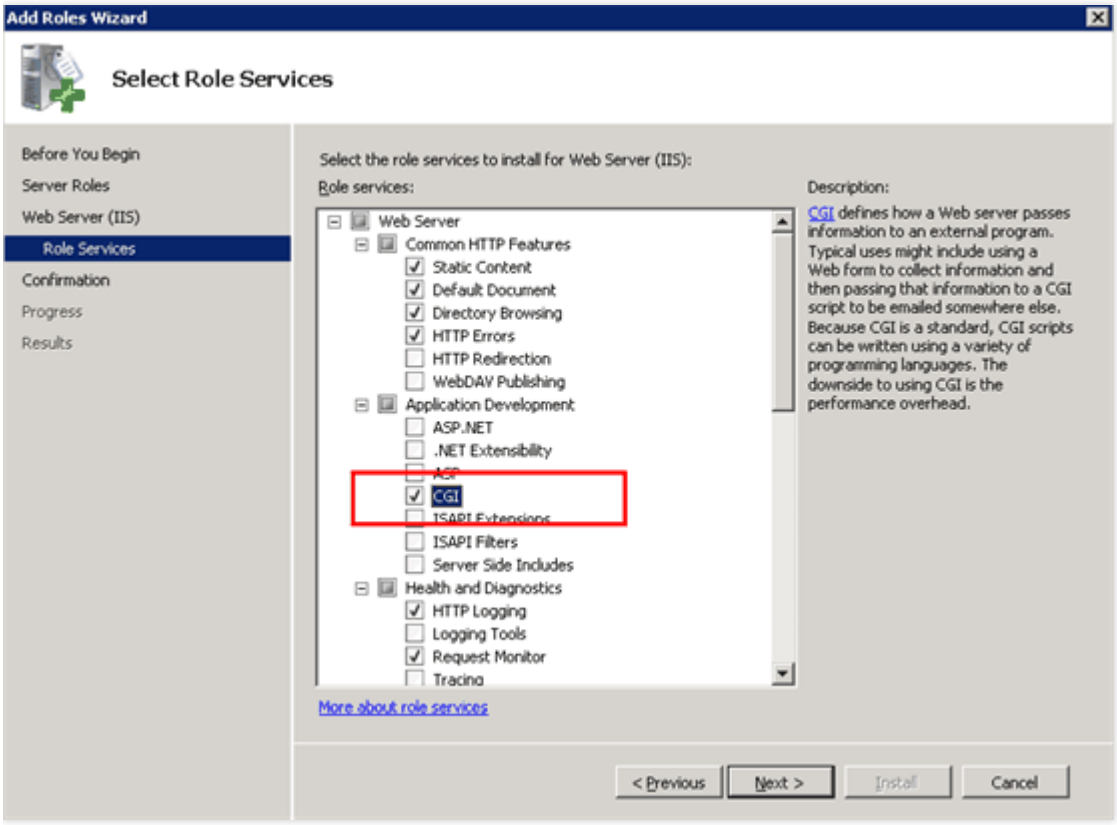
1) Click "Server Manager" in the "Management Tool" in the "Start" menu at the bottom left corner of Windows CVM to open the Server Manager interface.

2) Click "Add Roles and Features" to add server roles. In this case, select "Web Server (IIS)", as shown below:

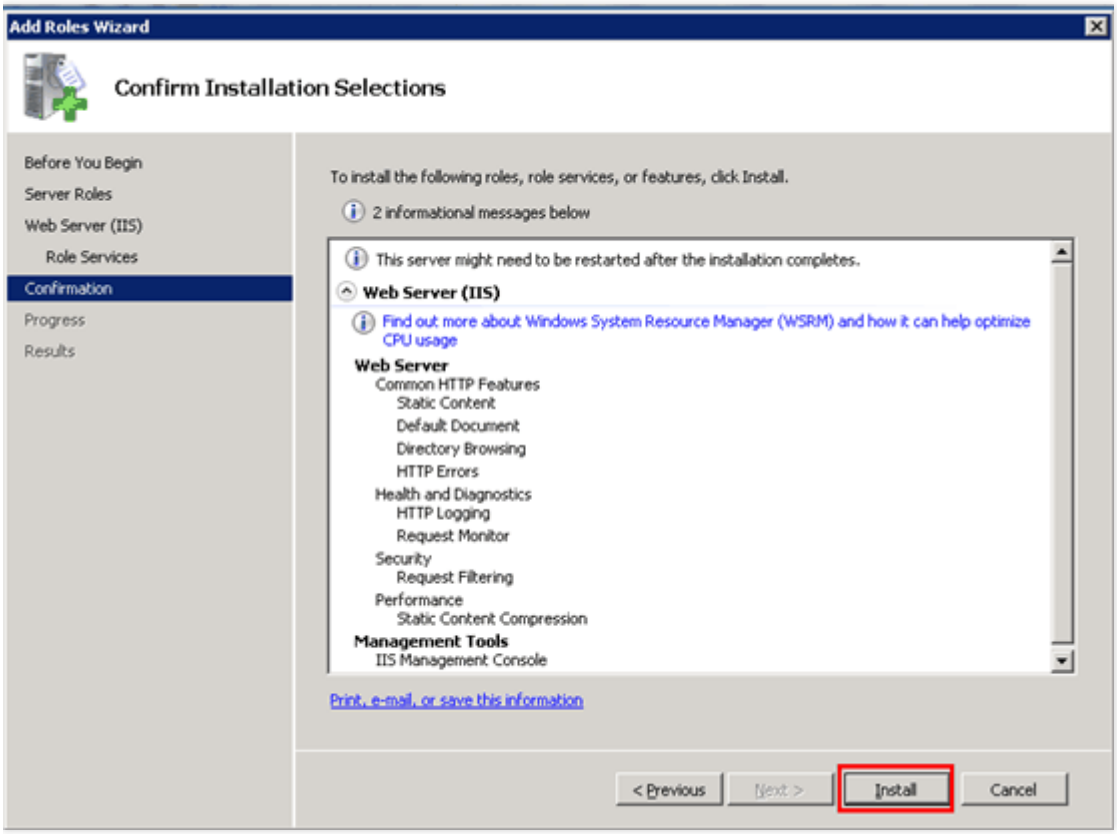




3) Click "Next". When selecting role services, check "CGI", as shown below:



4) After the settings are made, click "Install" to proceed with the installation:

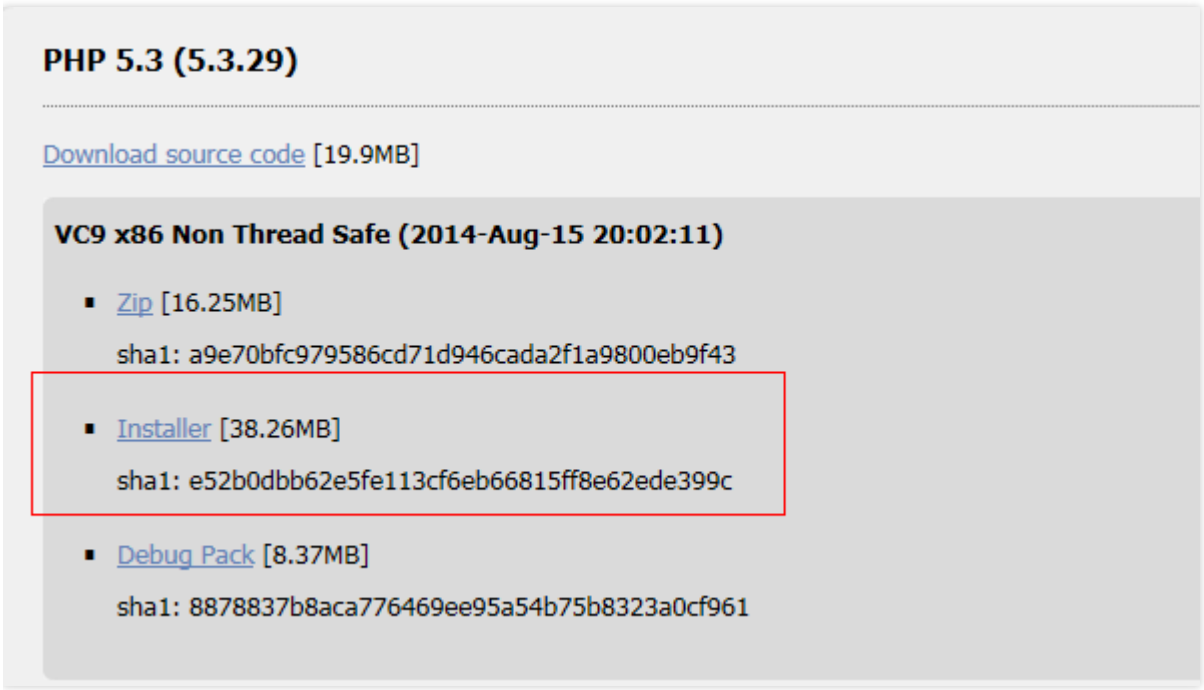


5) Access the public network IP of Windows CVM via browser to check whether the IIS service is running normally. The appearance of the following page indicates that IIS has been installed and configured successfully.

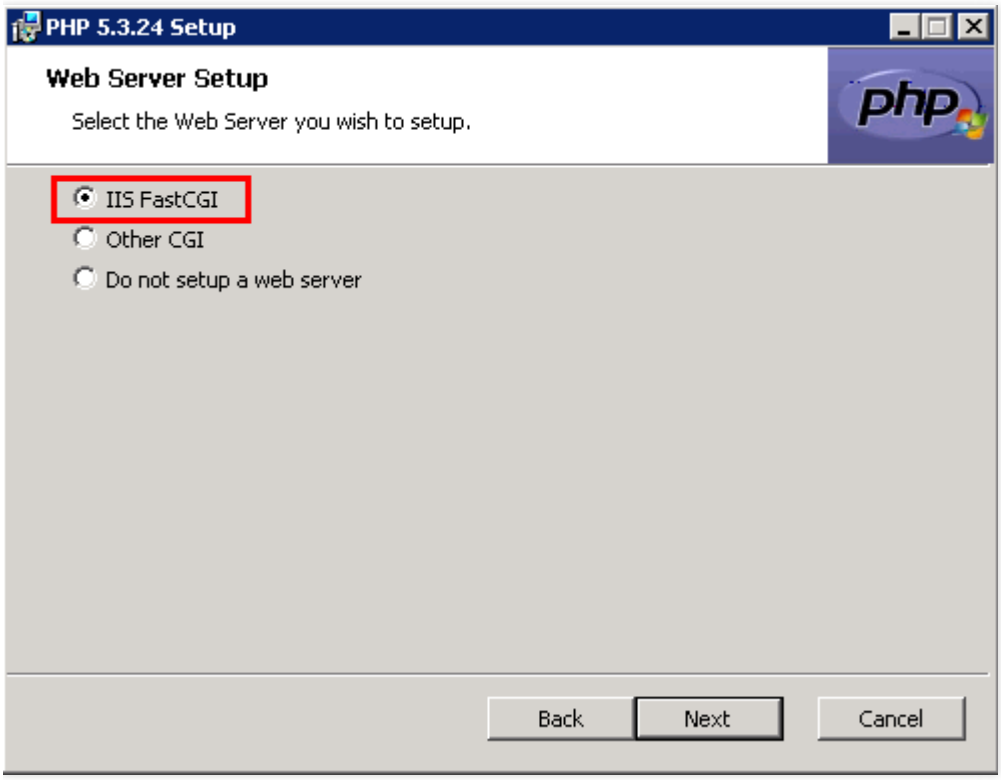
2. Installation and configuration of PHP

2.1. Installation of PHP 5.3 and earlier versions

1) Download the PHP installer (Download from: http://windows.php.net/download/), select the installer indicated in the following figure:

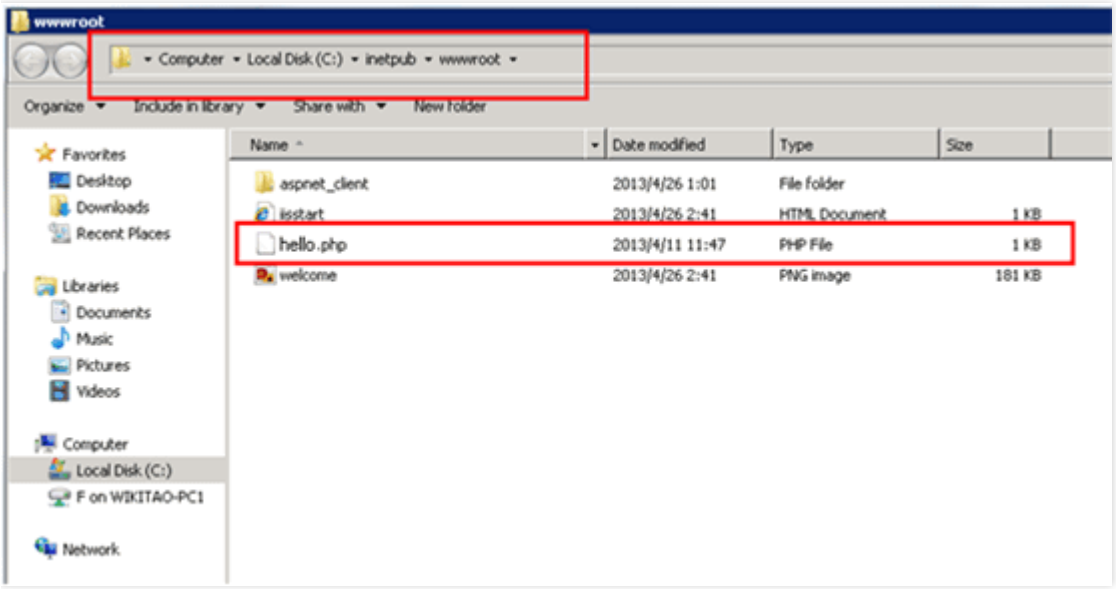


2) After the download, install PHP. When you need to select Web service, select "IIS FastCGI", as shown below:



3) Complete the installation of PHP under the guidance of installation interface.

4) Create a PHP file hello.php under C: / inetpub / wwwroot, as shown below:



The following content is written to the hello.php file:

```
Test Page";
echo "hello world";
?>
```

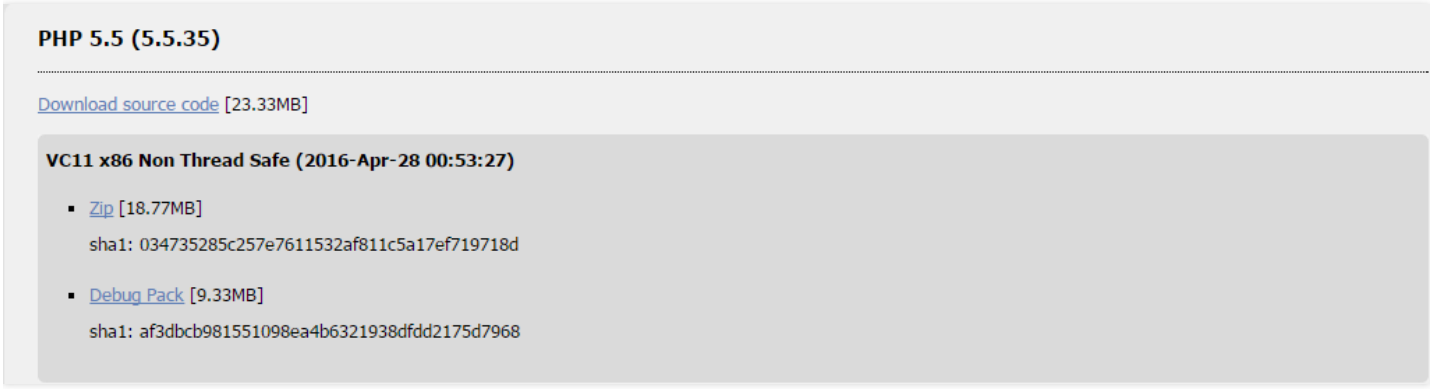
5) Access the public network IP of Windows CVM via browser to check whether the environment configuration has been completed successfully.

2.2. Installation of PHP versions above 5.3

For PHP versions above 5.3, the installer mode has been canceled, and the installation is only performed through zip file or debug pack. The following example shows the zip installation in Windows Server 2012R2 environment.

1) Download the PHP zip installer. Please note that you must select Non Thread Safe (NTS) x86 package when running under IIS. (If you have to select x64 package for PHP in Windows Server 32bit (x64), you cannot select IIS. In this case, you can use Apache as an alternative option)

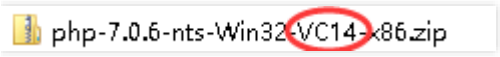
Select the installer as shown below:



2) The installation of PHP versions above 5.3 depends on Visual C ++ Redistributable Update. Download and install VC Update Installer according to the name of downloaded PHP installer by referring to the relations as shown in the following table:

PHP Installer Name	Download Link for Visual C ++ Redistributable Installer
Php-xxx-nts-Win32-VC14-x86.zip	Visual C ++ Redistributable for Visual Studio 2015
Php-xxx-nts-Win32-VC11-x86.zip	Visual C ++ Redistributable for Visual Studio 2012 Update 4
Php-xxx-nts-Win32-VC9-x86.zip	Microsoft Visual C ++ 2008 SP1 Redistributable Package (x86)

For example, if the downloaded PHP installer is the one shown as below,



then download the installer for VS2015 version based on the relation indicated in the first row, and download and install the .exe file.

3) Unzip the PHP zip installer (in this case, extract to C:\PHP), copy php.ini-production and rename it to php.ini.

4) Click "Server Manager" - "IIS"; On the local IIS, right-click and select IIS Manager.

Click on the host name (IP) on the left to go to the home page, then double-click "Handler Mappings".

Click "Add Module Mappings" button on the right, fill in the following information in the pop-up box, and click "OK" to save.

If you are unable to select php-cgi.exe as the executable file, please change the file name extension of the selected file to .exe.

5) Click on the host name (IP) on the left to return to the home page, then double-click "Default Document".

Click "Add" button on the right to add the default document with the name of index.php.

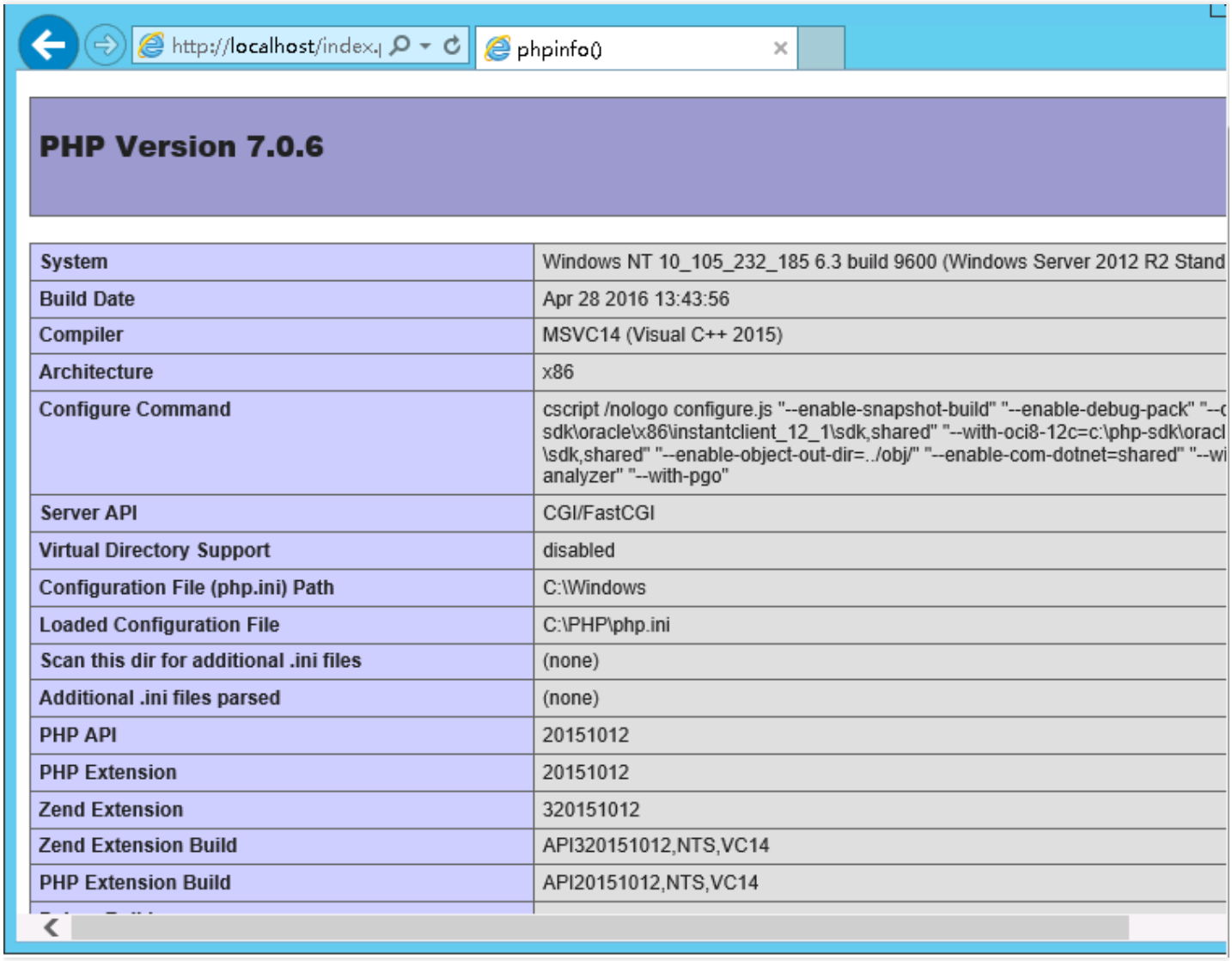
6) Click on the host name (IP) on the left to return to the home page, then double-click "FastCGI Settings".

Select the path, click the "Edit" button on the right, then in the "Monitor the Changes Made to File", select the php.ini path.

7) Create a PHP file index.php under C:\inetpub\wwwroot, to which the following content is written:

```
<?php

phpinfo ();

?>
```

Save, visit http: //localhost/index.php within from the CVM to verify whether PHP has been installed successfully:



Install and Configure PHP

Last updated：2018-08-06 15:44:27

This document introduces the PHP configuration of Windows CVM. For more information, please see Installation of PHP versions above 5.3 and Installation of PHP 5.3 and earlier as needed.

Prerequisites

To configure PHP in Windows CVMs, you need to add and install IIS role. For more information, please see Installing and Configuring IIS.

Installation of PHP Versions above 5.3

For PHP versions above 5.3, the installer mode has been canceled, and the installation is only performed through zip file or debug pack. The following example shows the zip installation in Windows Server 2012 R2 environment.

Downloading File

1. Download the PHP zip installer from the CVM (download URL: http://windows.php.net/download/).

> Note:
> You must select Non Thread Safe (NTS) x86 package when running under IIS. If you have to select x64 package for PHP in Windows Server 32bit (x64), you cannot select IIS. In this case, you can use Apache as an alternative option.

Select the installer as shown below:

PHP 7.0 (7.0.6)

Download source code [24.08MB]

VC14 x86 Non Thread Safe (2016-Apr-29 00:38:17)

- Zip [20.93MB]
  sha1: cf0548e3f4def7e22f140dab243709b0676077aa
- Debug Pack [14.65MB]
  sha1: d17bfc88bda7a716dde0dad0e11424b7588ac0db

PHP 5.6 (5.6.21)

Download source code [24.15MB]

VC11 x86 Non Thread Safe (2016-Apr-28 06:19:34)

- Zip [20.29MB]
  sha1: 46f159ee7be2307aca4d8d5f80034f11c7fff5d2
- Debug Pack [9.68MB]
  sha1: 6aba9b880ddc8c5c6ed73559bfbb63649d2c82f9

PHP 5.5 (5.5.35)

Download source code [23.33MB]

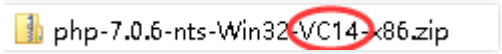
VC11 x86 Non Thread Safe (2016-Apr-28 00:53:27)

- Zip [18.77MB]
  sha1: 034735285c257e7611532af811c5a17ef719718d
- Debug Pack [9.33MB]
  sha1: af3dbcb981551098ea4b6321938dfdd2175d7968

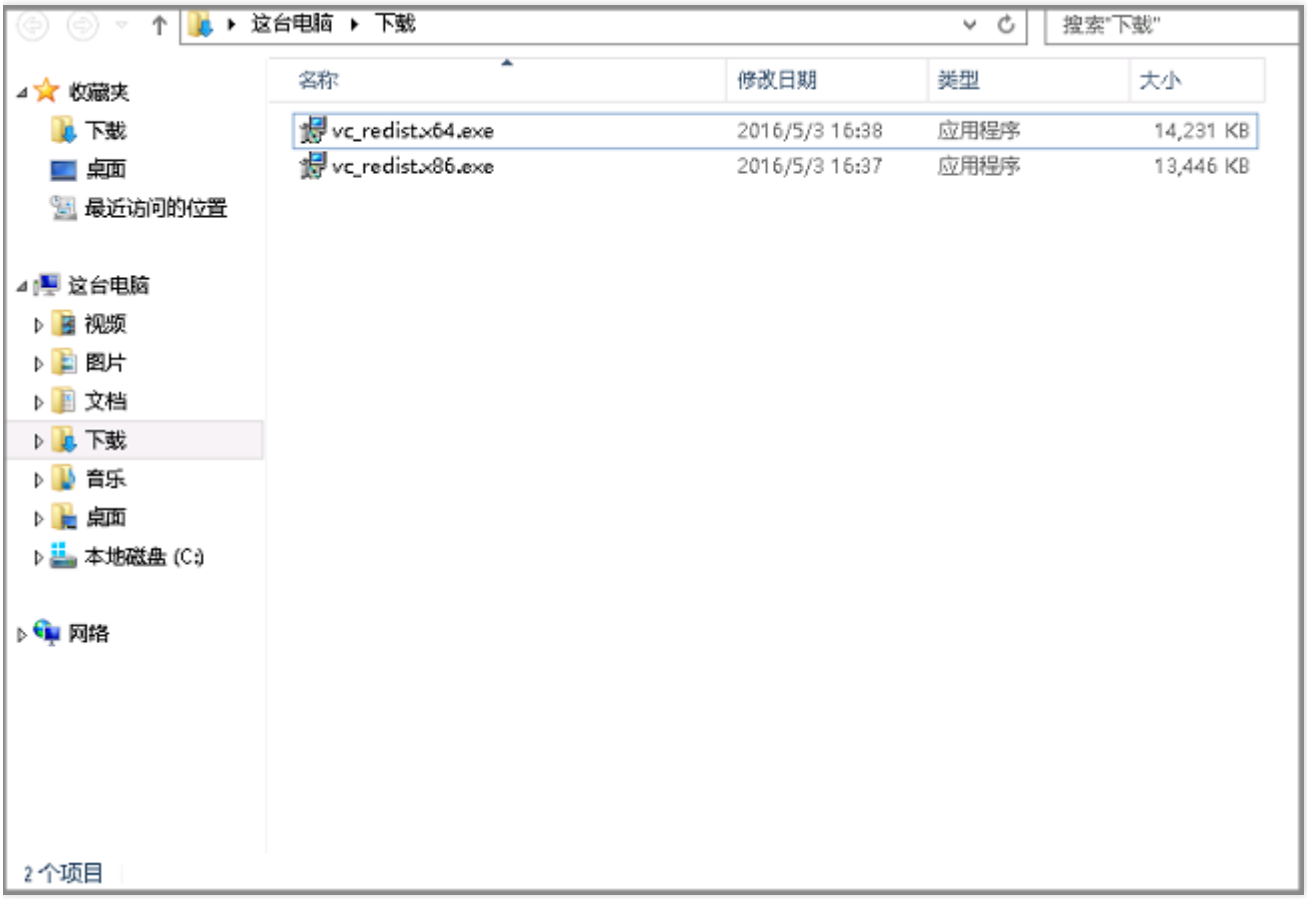
2. The installation of PHP versions above 5.3 is dependent on Visual C++ Redistributable Update. Download and install VC Update Installer according to the name of downloaded PHP installer by referring to the relations as shown in the following table:

PHP Installer Name	Download Link for Visual C++ Redistributable Installer
php-x.x.x-nts-Win32-VC14-x86.zip	Visual C++ Redistributable for Visual Studio 2015
php-x.x.x-nts-Win32-VC11-x86.zip	Visual C++ Redistributable for Visual Studio 2012 Update 4
php-x.x.x-nts-Win32-VC9-x86.zip	Microsoft Visual C++ 2008 SP1 Redistributable Package (x86)

If the downloaded PHP installer is as shown below:

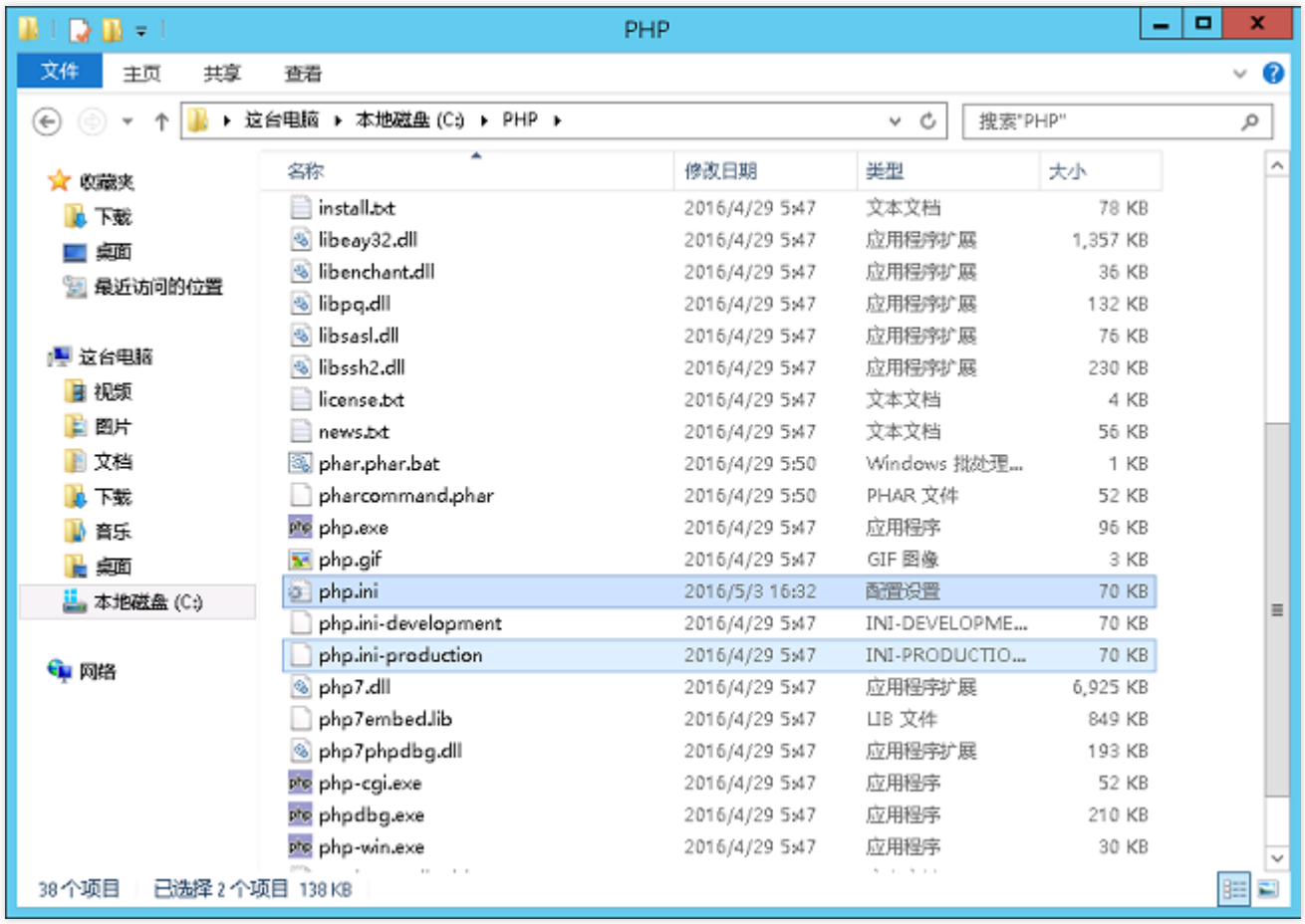


then download the installer for VS 2015 version based on the relation indicated in the first row, and download and install the following two `.exe` files:

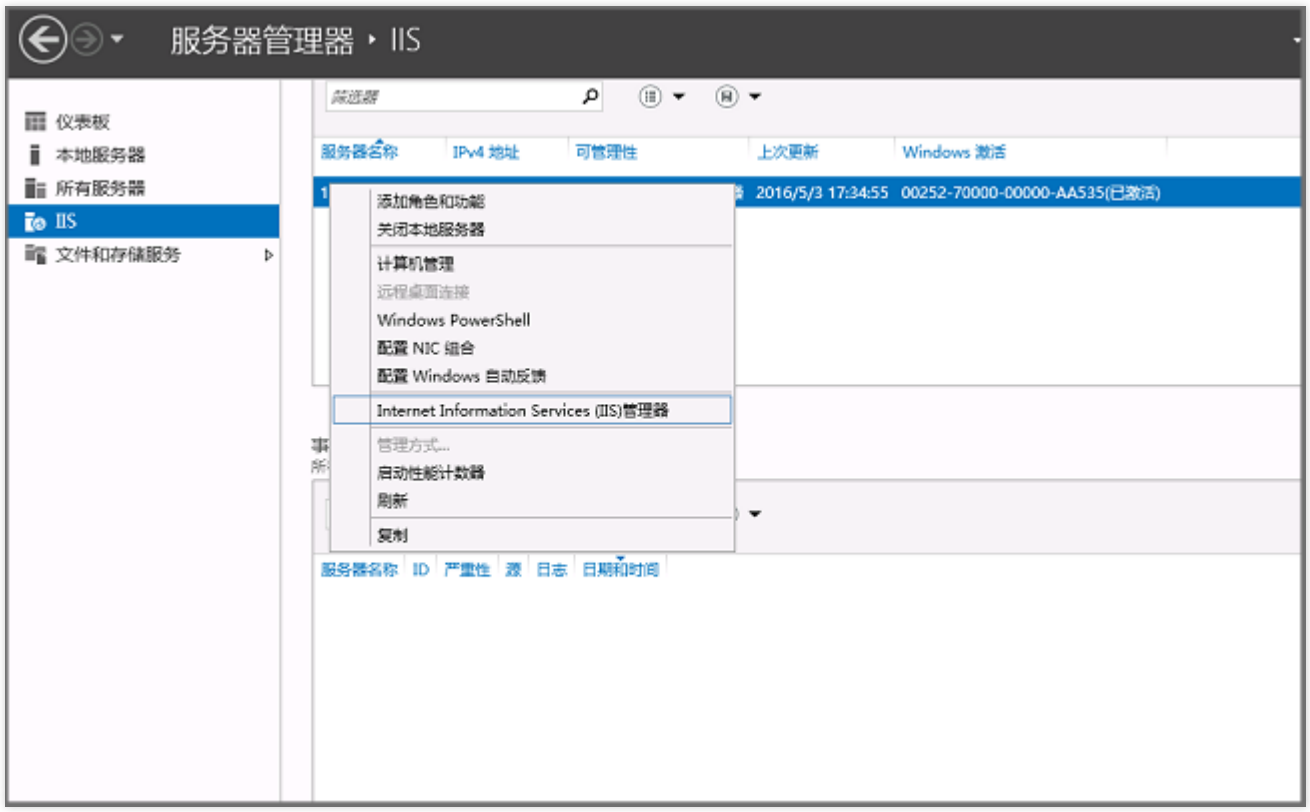


Installation and configuration

1. Unzip the PHP zip installer (in this case, extract to `C:\PHP`), copy `php.ini-production` and rename it to `php.ini`, as shown below:

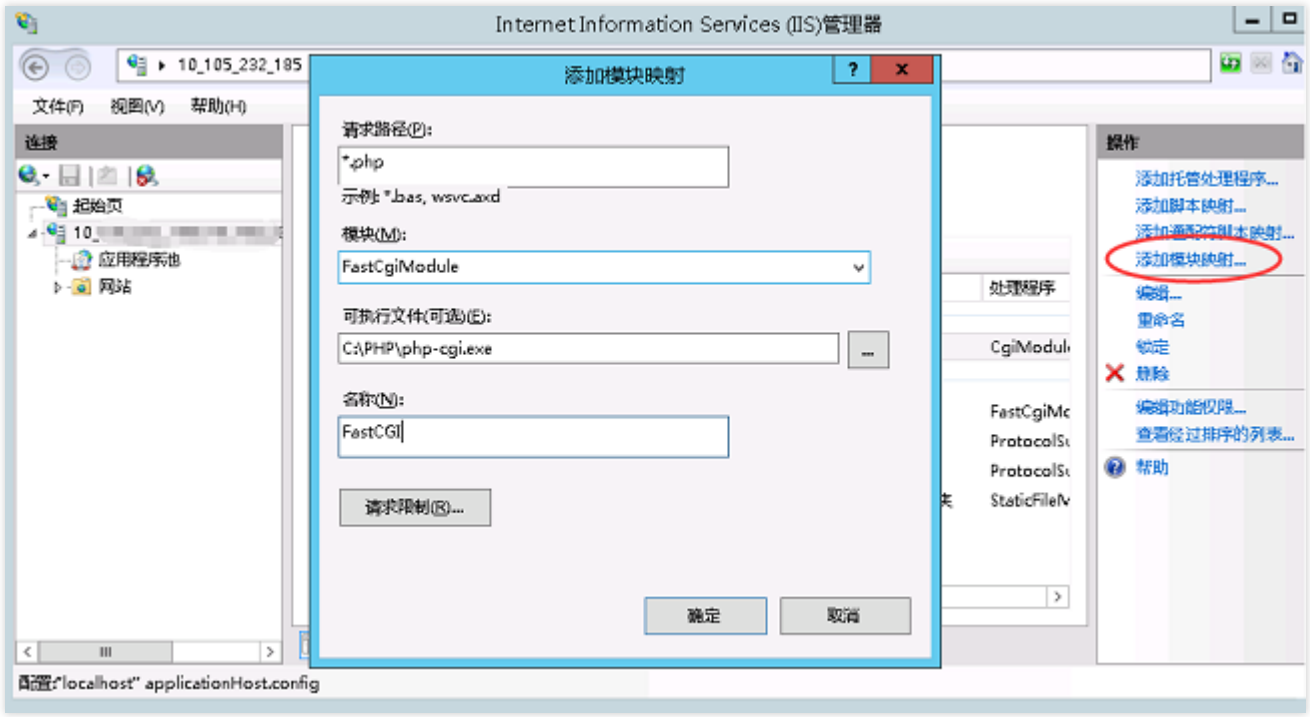


2. Click **Server Manager** -> **IIS**; On the local IIS, right-click and select **IIS Manager**:



Click on the host name (IP) on the left to go to the home page, and then double-click **Handler Mappings**:

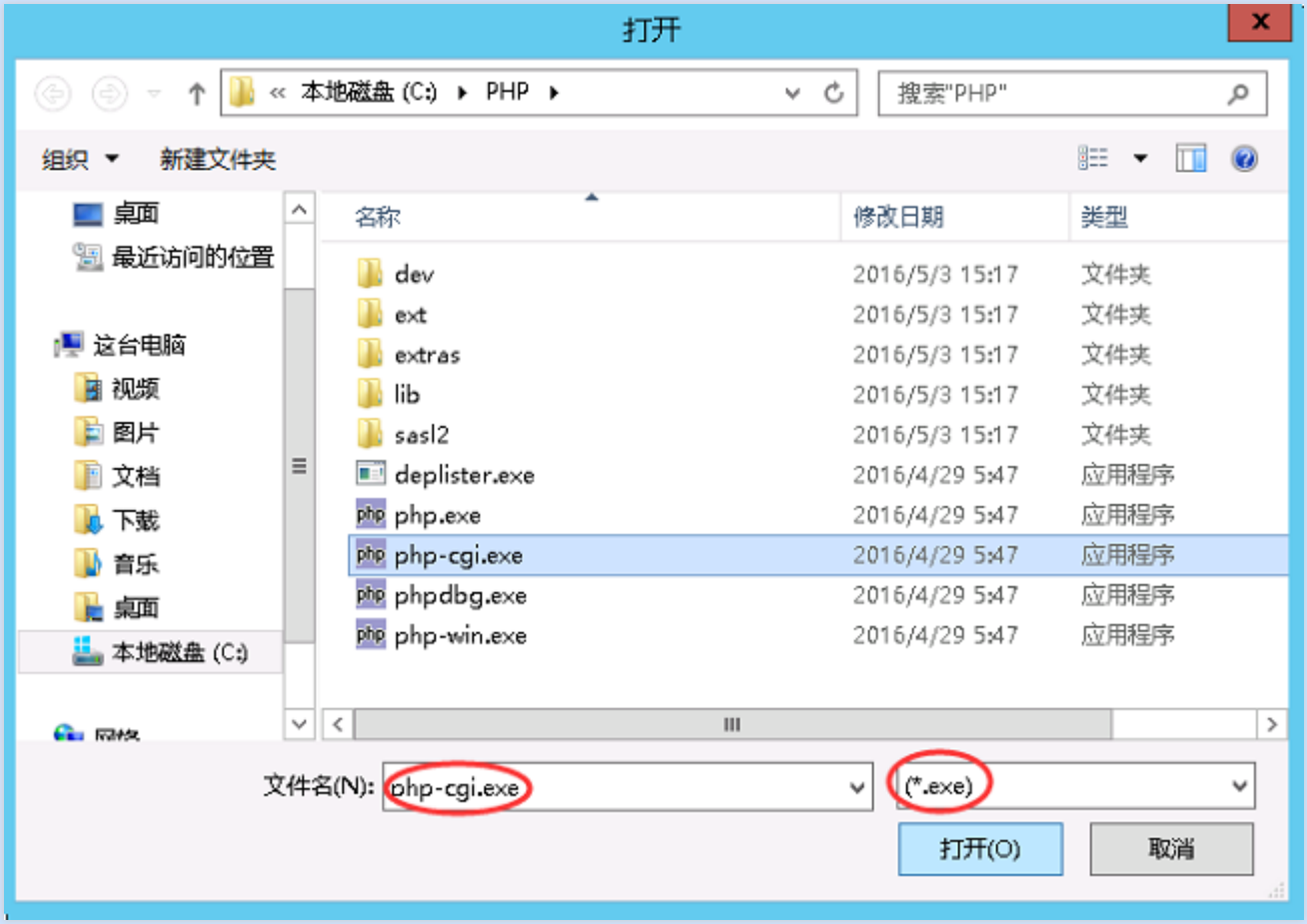
Click **Add Module Mappings** on the right, enter the following information in the pop-up box, and click **OK** to save:



Note:

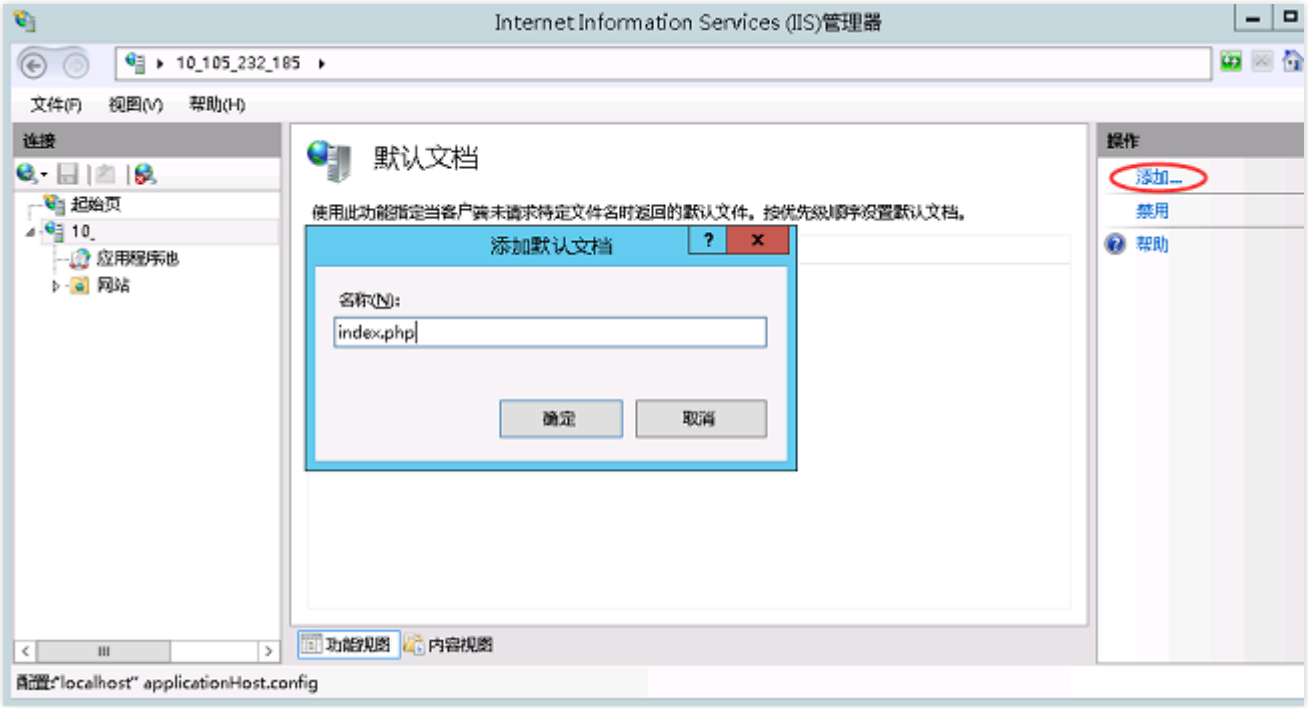
If you are unable to select `php-cgi.exe` as the executable file, please change the filename

extension of the selected file to .exe:



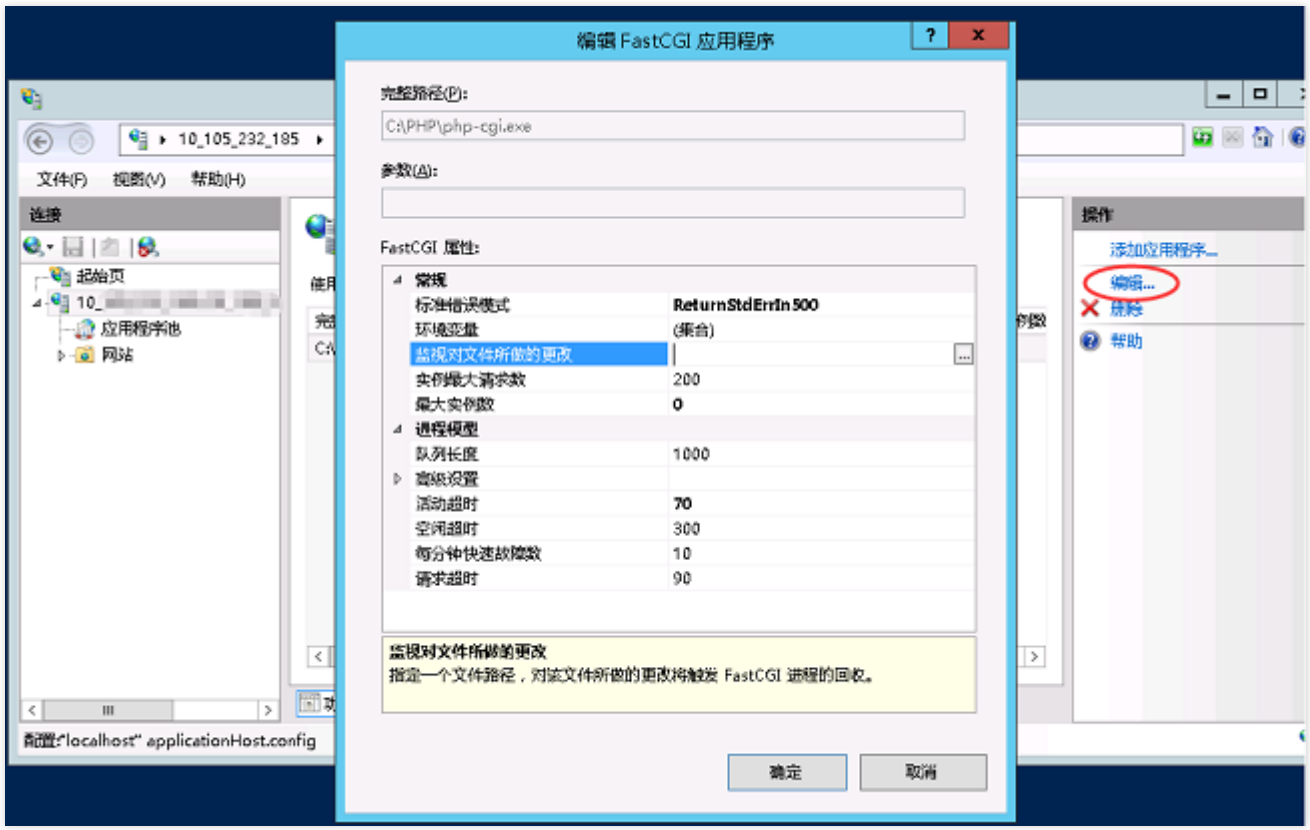
3. Click the server IP on the left to go to the home page and double-click **Default Documents**:

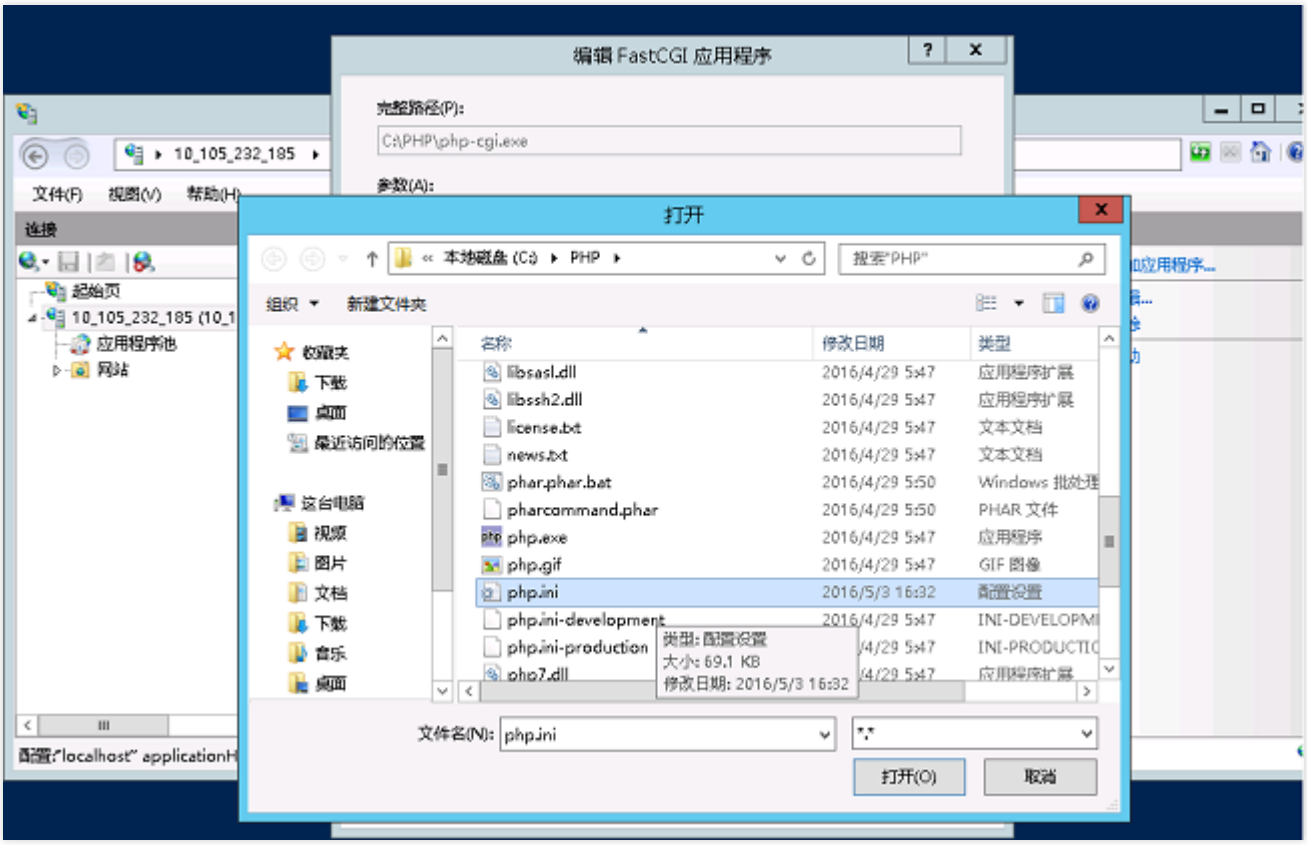
Click **Add** on the right to add a default document named `index.php` :



4. Click the server IP on the left to go to the home page and double-click **FastCGI Settings**:

Click **Edit** on the right, and select `php.ini` path in **Monitor the Changes Made to File**:



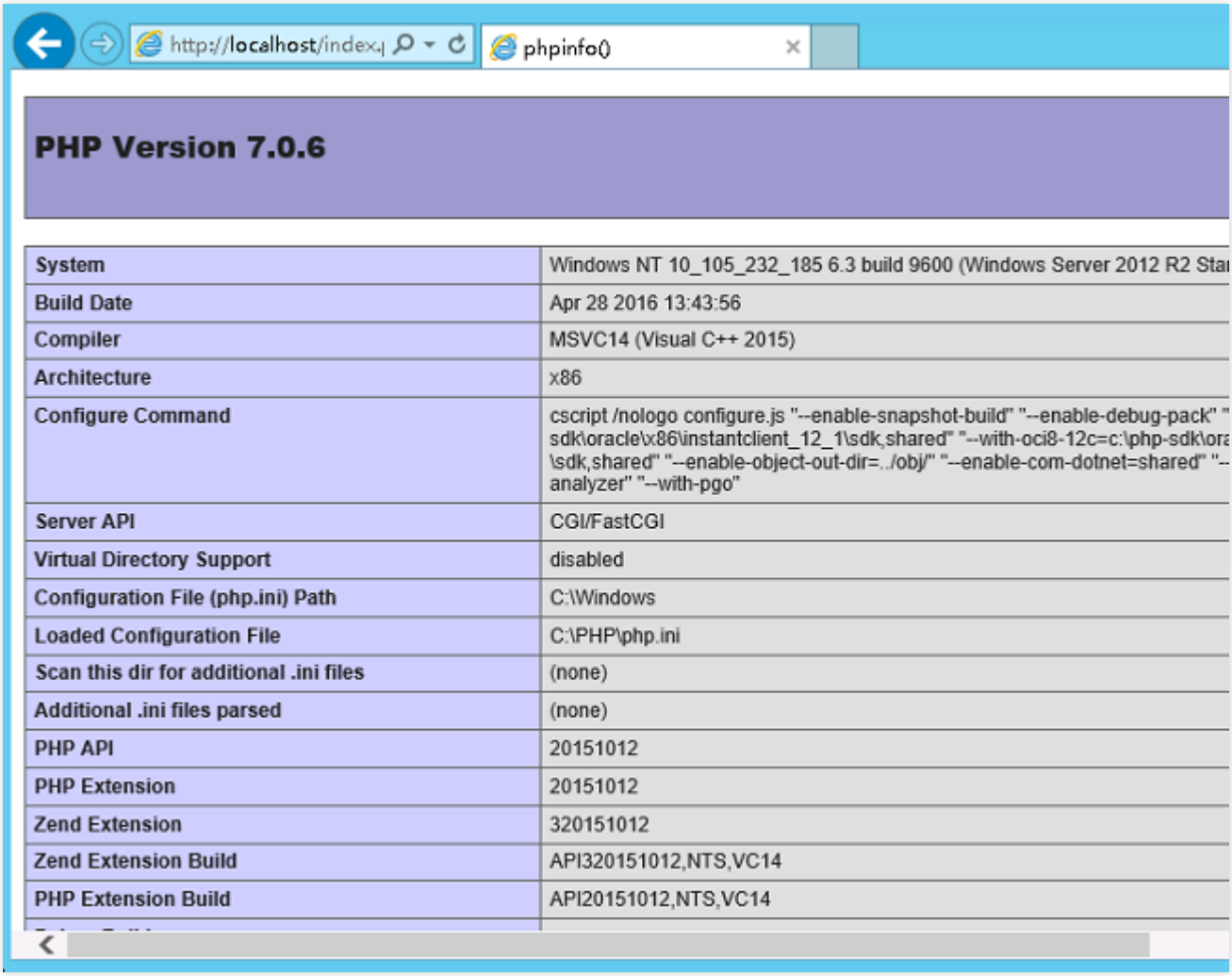


5. Create a PHP file `index.php` under `C:\inetpub\wwwroot` and write the following:

```
<?php
phpinfo();
?>
```

6. Visit `http://localhost/index.php` in the browser on CVM and check whether the environment has been configured successfully. The appearance of the following page indicates that the configuration has

been completed successfully:

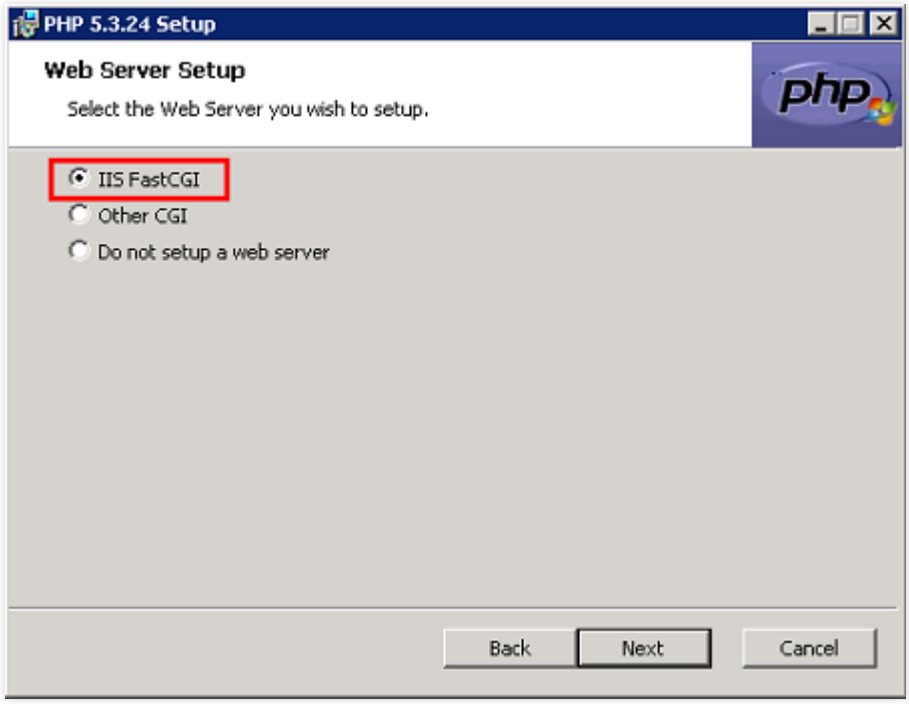


Installation of PHP 5.3 and earlier versions

Note:
PHP 5.3 and earlier versions are no longer available on the official download address http://windows.php.net/download/. To use these versions, download them locally and upload them to the CVM or search on the CVM network. For more information on uploading files, please see here.

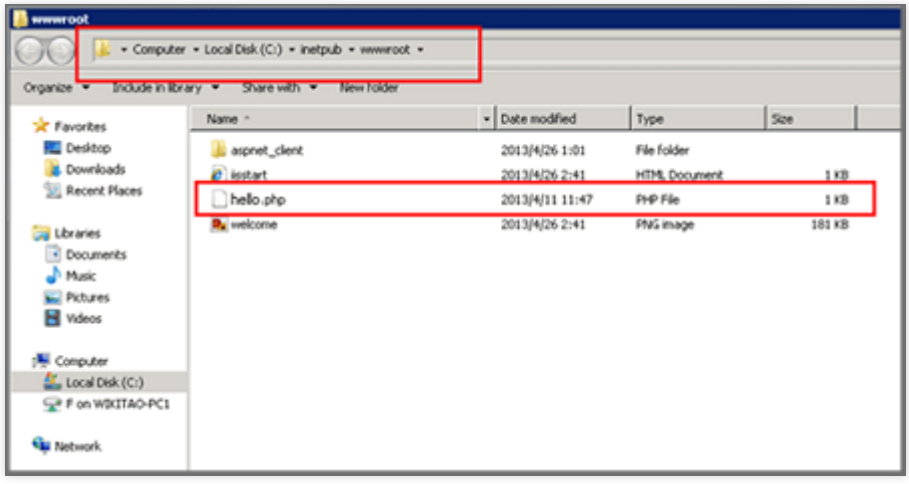
1. Open PHP installer in the CVM.

2. Select **IIS FastCGI** in **Web Server Setup**, as shown below:



3. Complete the installation of PHP under the guidance of installation interface.

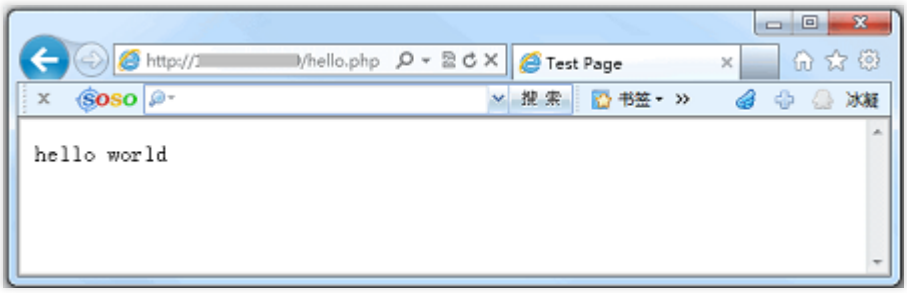
4. Create a PHP file `hello.php` under `C:/inetpub/wwwroot`, as shown below:



The following content is written to the `hello.php` file:

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

5. Access the public network IP of Windows CVM via browser to check whether the environment configuration has been completed successfully. The appearance of the following page indicates that the configuration has been completed successfully:



Install and Build MySQL

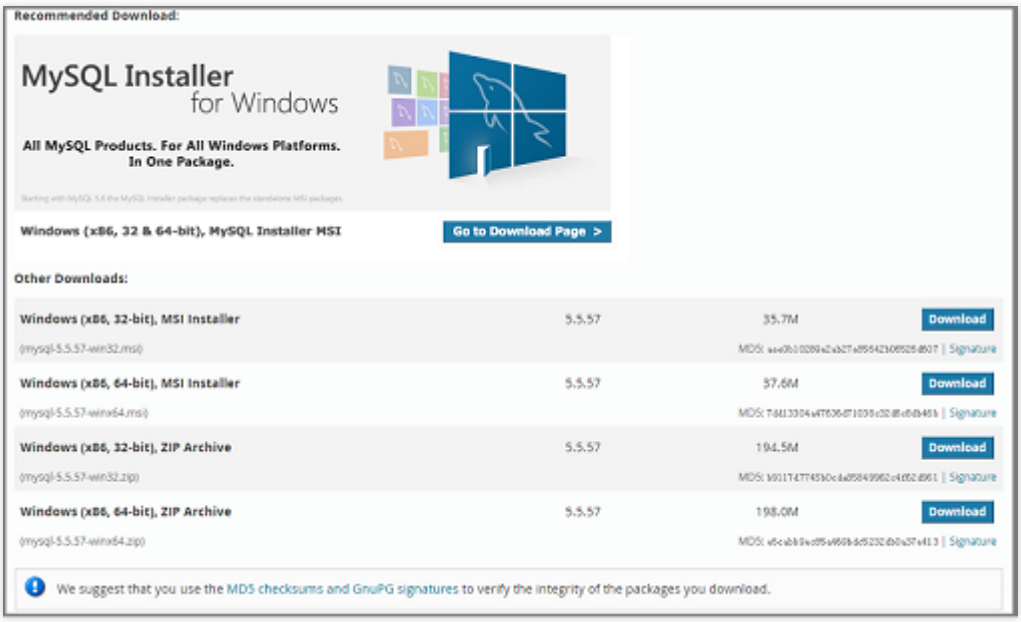
Last updated：2018-08-06 15:43:34

This document uses Windows Server 2012 R2 as an example to introduce how to build MySQL 5.5.

SQL Server database is frequently used in Windows system. You need to grand authorization for SQL Server because it is not for free. You can also purchase CDB instances for Tencent Cloud SQLServer database.

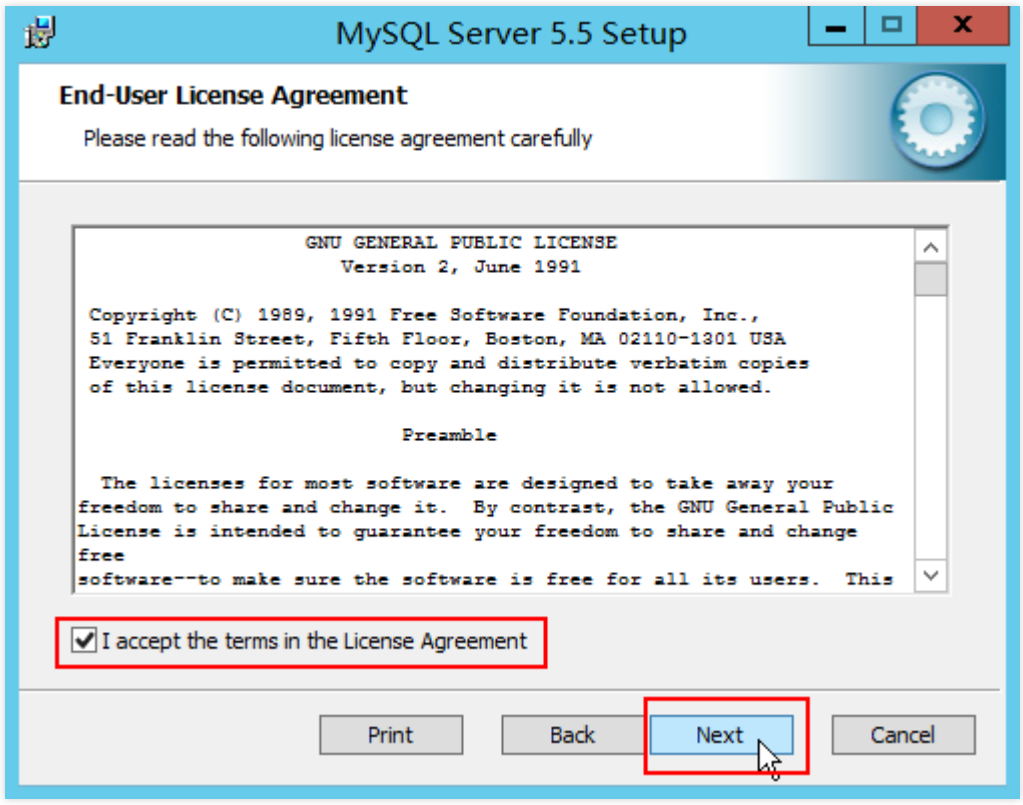
Step 1: Download MySQL installer

Open the browser on the CVM and enter the download URL:https://dev.mysql.com/downloads/mysql/5.5.html#downloads

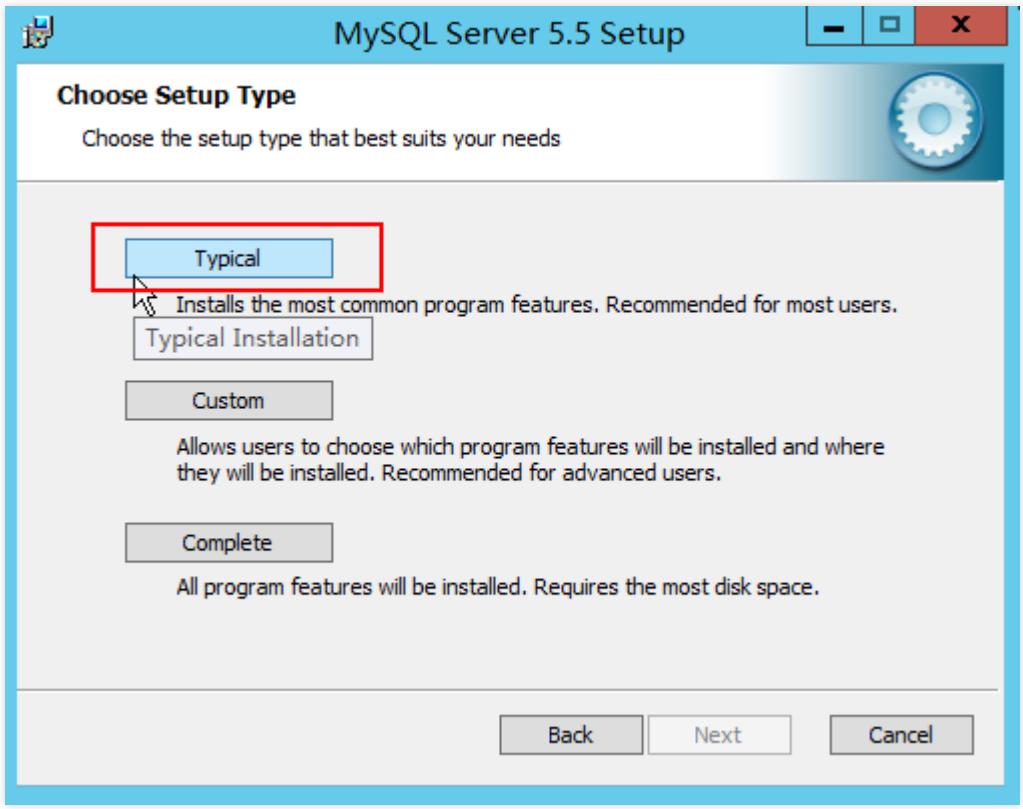


Step 2: Install the application

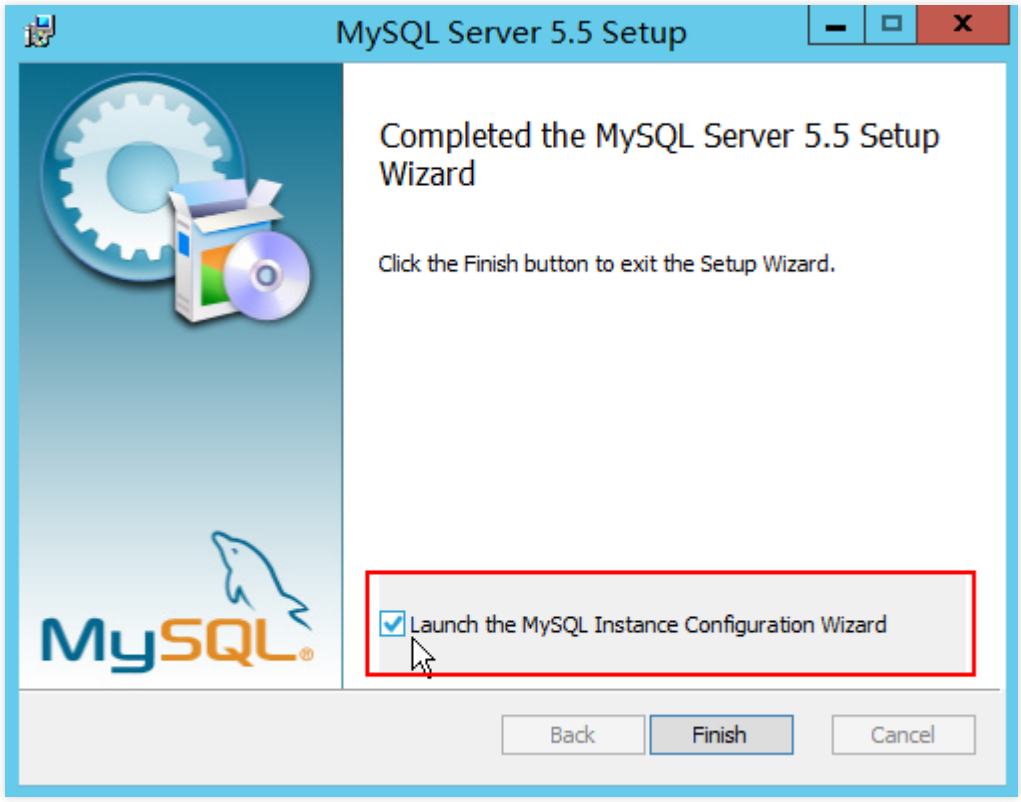
1. Run the installation program. Click **Next** and select **I accept the terms in the License Agreement**.



2. Select **Typical** in **Choose Setup Type**.



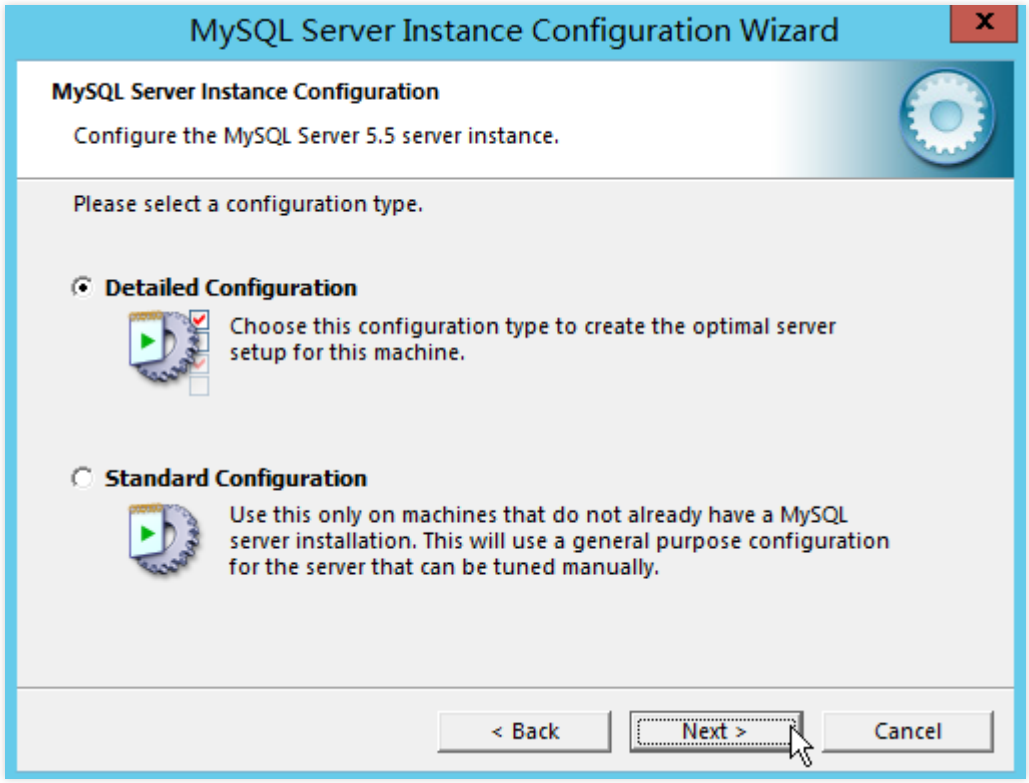
3. Select **Launch the MySQL Instance Configuration Wizard**.



Step 3: Configure MySQL

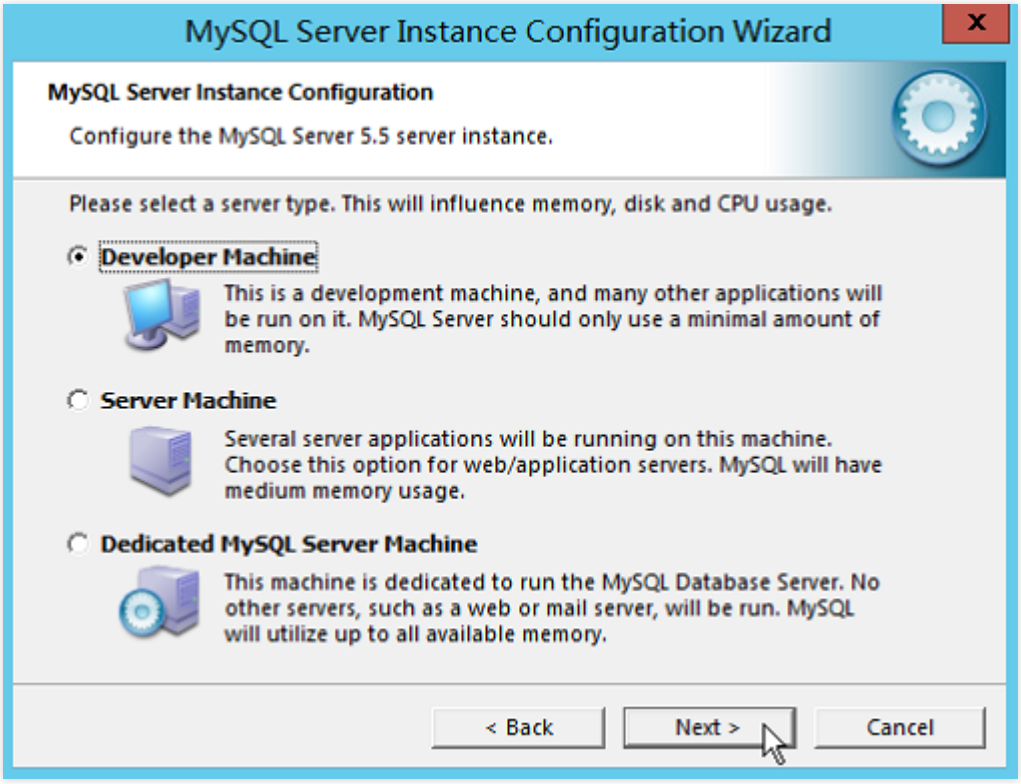
1. Configure the type of MySQL. Here we use Detailed Configuration as an example.
   - Detailed Configuration is suitable for advanced users that need to have finer control of the CVM configurations.
   - Standard Configuration is suitable for new users that want to launch MySQL quickly without considering the CVM configurations.

> **Note:**
> Standard Configuration may be incompatible with your operating system. Detailed Configuration is recommended.



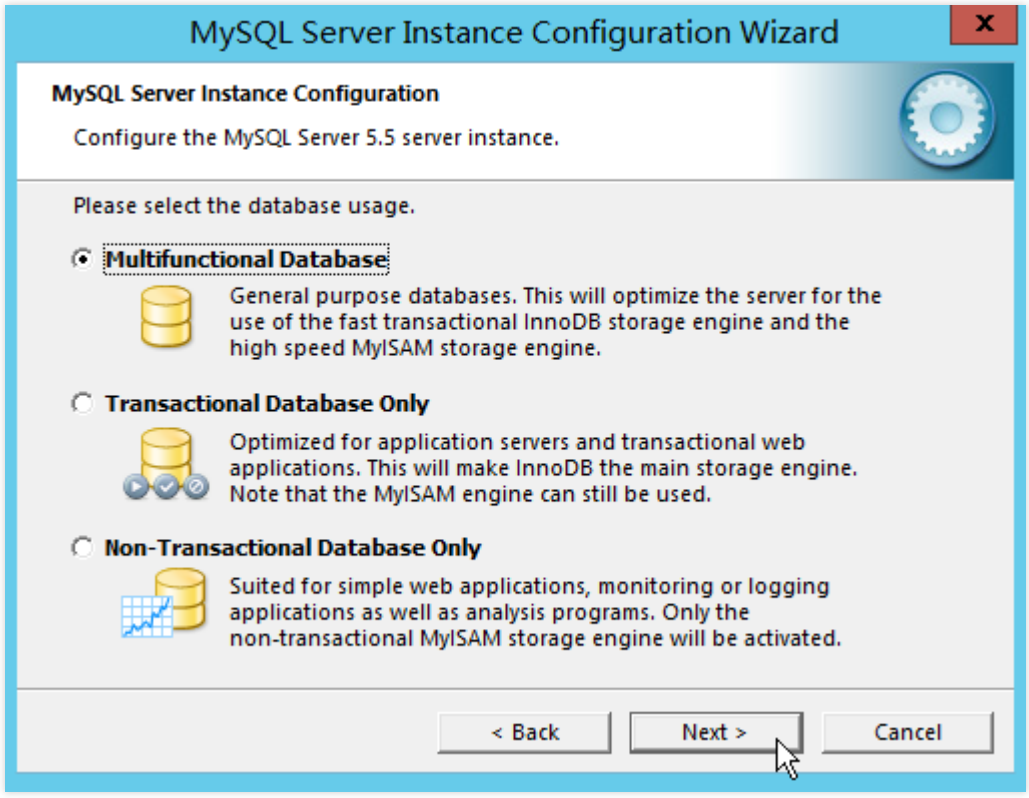
2. Configure the type of MySQL server. Here we use Developer Machine as an example.
   - Developer Machine represents a typical personal desktop workstation. When multiple desktop applications are running at the same time, the MySQL server is configured to use minimal system resources.

- Server Machine is a type of server on which MySQL server can run with other applications such as FTP, email and Web servers. The MySQL server is configured to use a moderate portion of the system resources.
- Dedicated MySQL Server Machine is a type of server on which only MySQL server can run. The MySQL server is configured to use all available system resources.

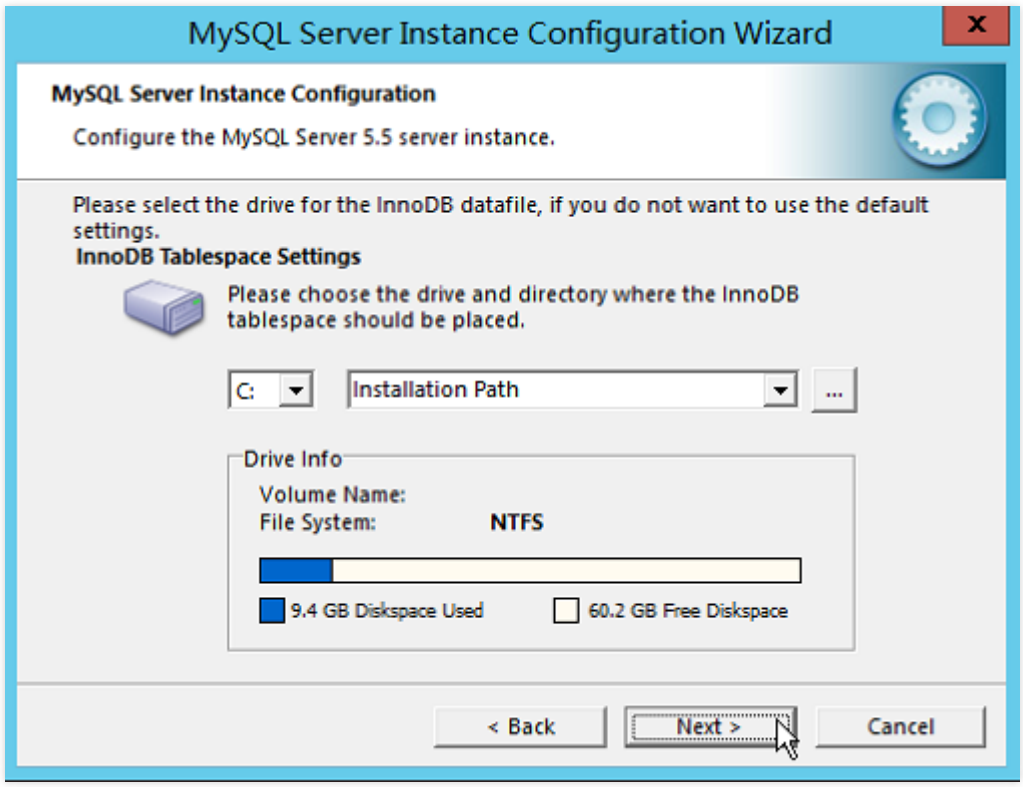


3. Configure MySQL database. Here we use Multifunctional Database as an example.

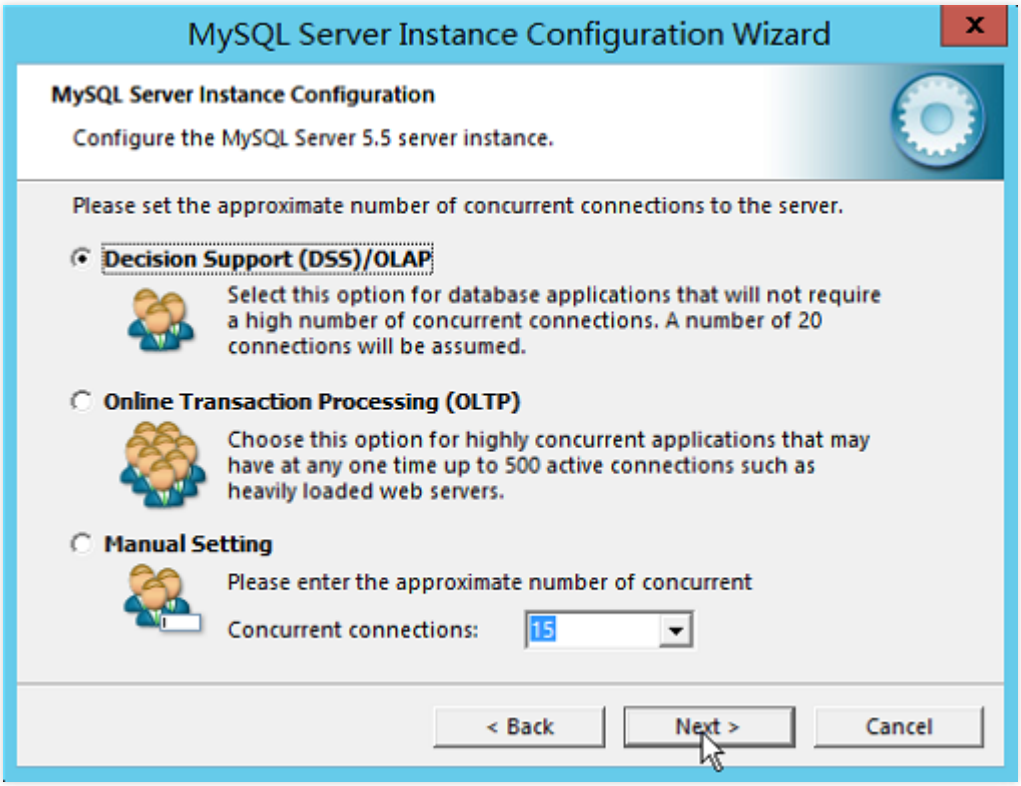
- Multifunctional Database uses InnoDB and MyISAM storage engines simultaneously, and allocates resources to them equally. You are recommended to select this option if you often use two storage engines simultaneously.
- Transactional Database Only uses InnoDB and MyISAM storage engines simultaneously, and allocates most server resources to InnoDB storage engine. You are recommended to select this option if you use InnoDB frequently and use MyISAM occasionally.
- Non-Transactional Database Only does not use InnoDB storage engine, and allocates all server resources to MyISAM storage engine. You are recommended to select this option if you do not use InnoDB.



4. Configure the InnoDB tablespace for MySQL. Choose default configuration here.

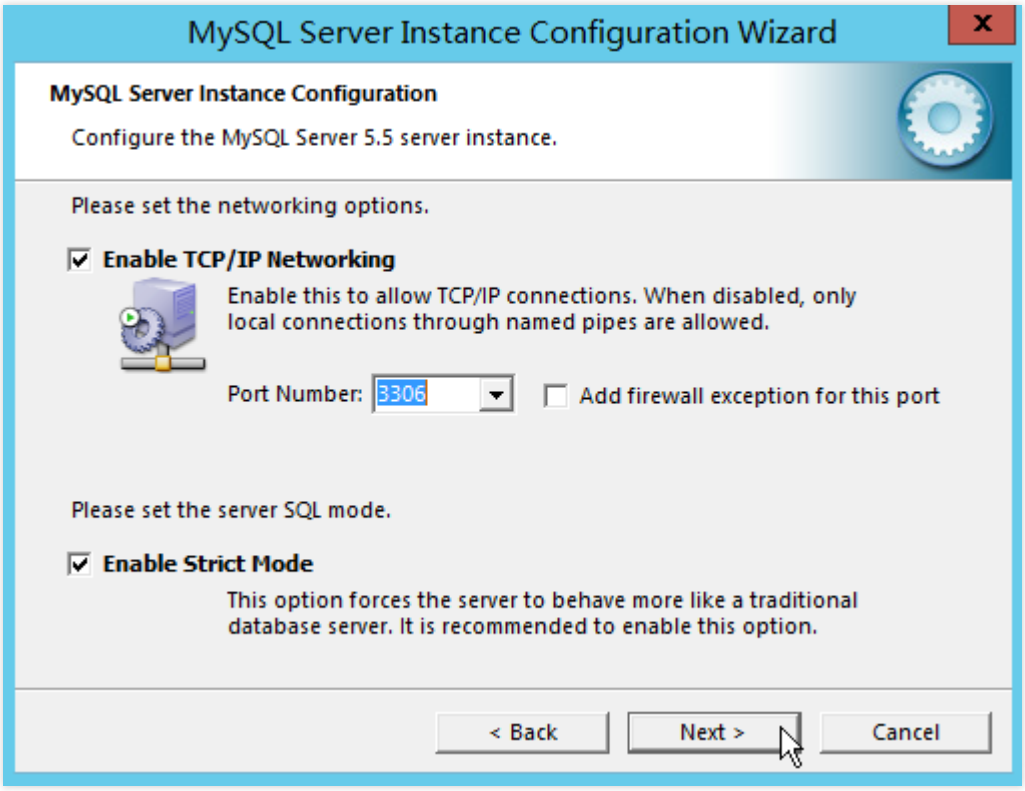


5. Configure concurrent connection for MySQL. Here we use Decision Support as an example.
   - Decision Support is suitable for situations that do not require a large number of concurrent connections.
   - Online Transaction Processing is suitable when a large number of concurrent connections are required.
   - Manual Setting is suitable when you need to configure the maximum number of concurrent connections manually.



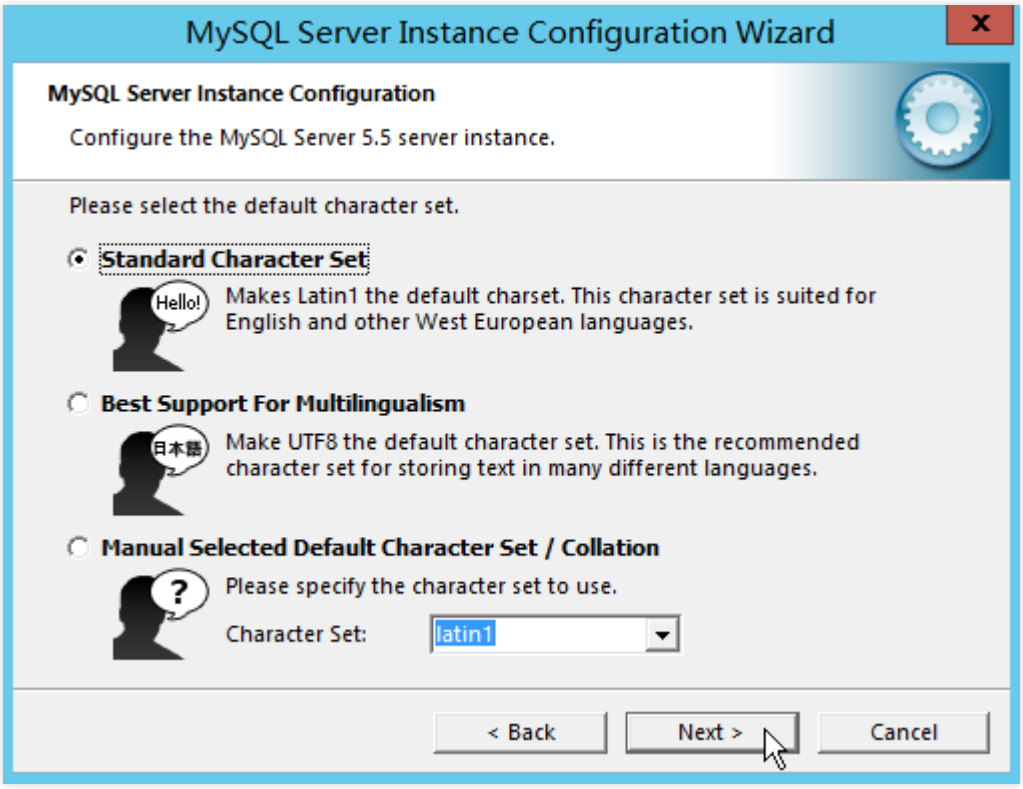
6. Configure the network options of MySQL. You can enable or disable TCP/IP network and configure the port number for MySQL server connection.

> **Notes:**
> TCP/IP network is enabled by default.
> Port 3306 is used by default.

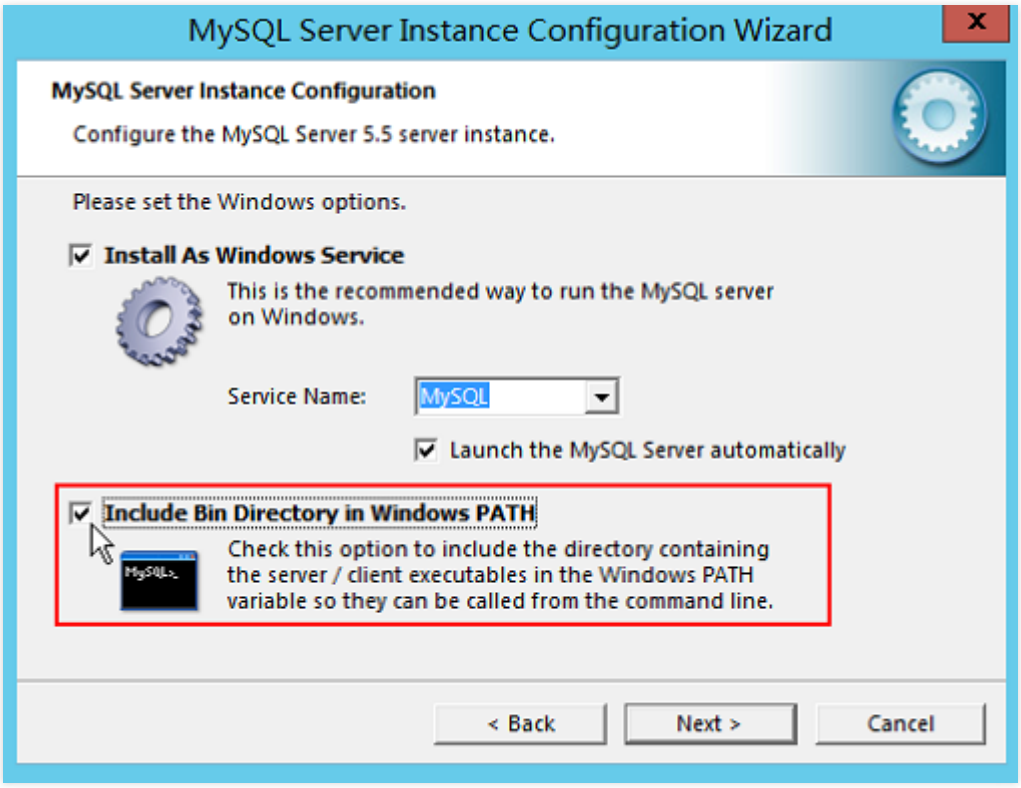


7. Configure MySQL character set. Here we use Standard Character Set as an example.

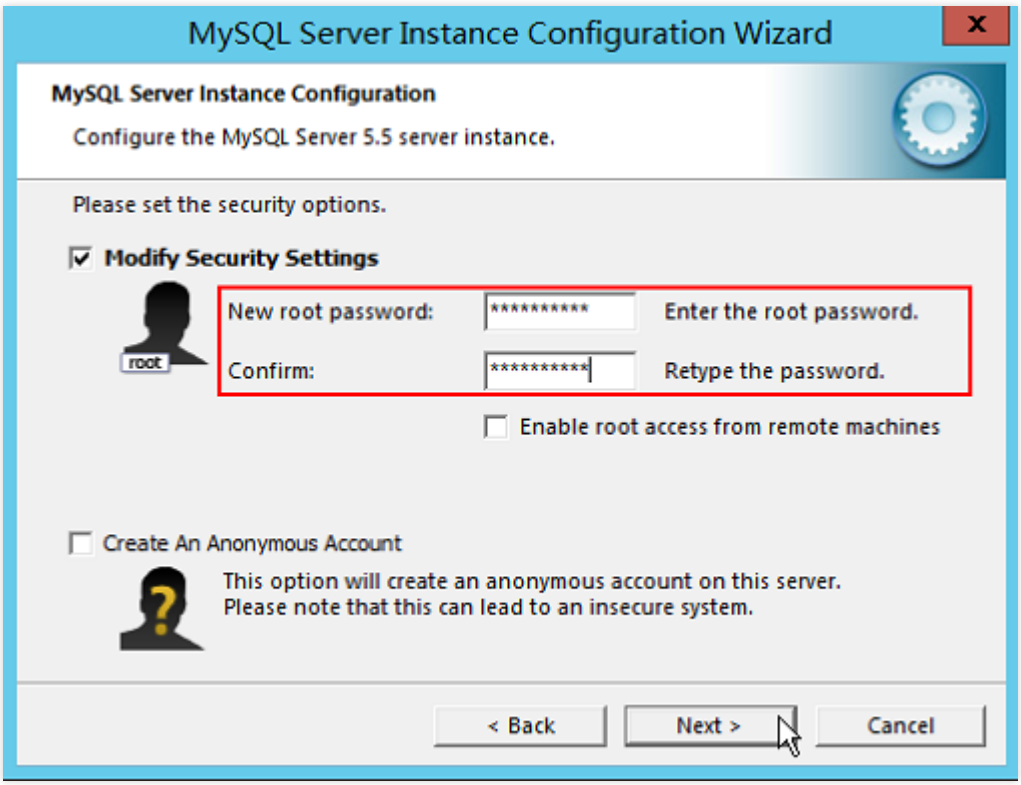
- Standard Character Set uses Latin1 as the default server character set.
- Best Support For Multilingualism uses UTF8 as the server character set.
- For Manual Selected Default Character Set/Collation, select the character set in the drop-down box as needed.



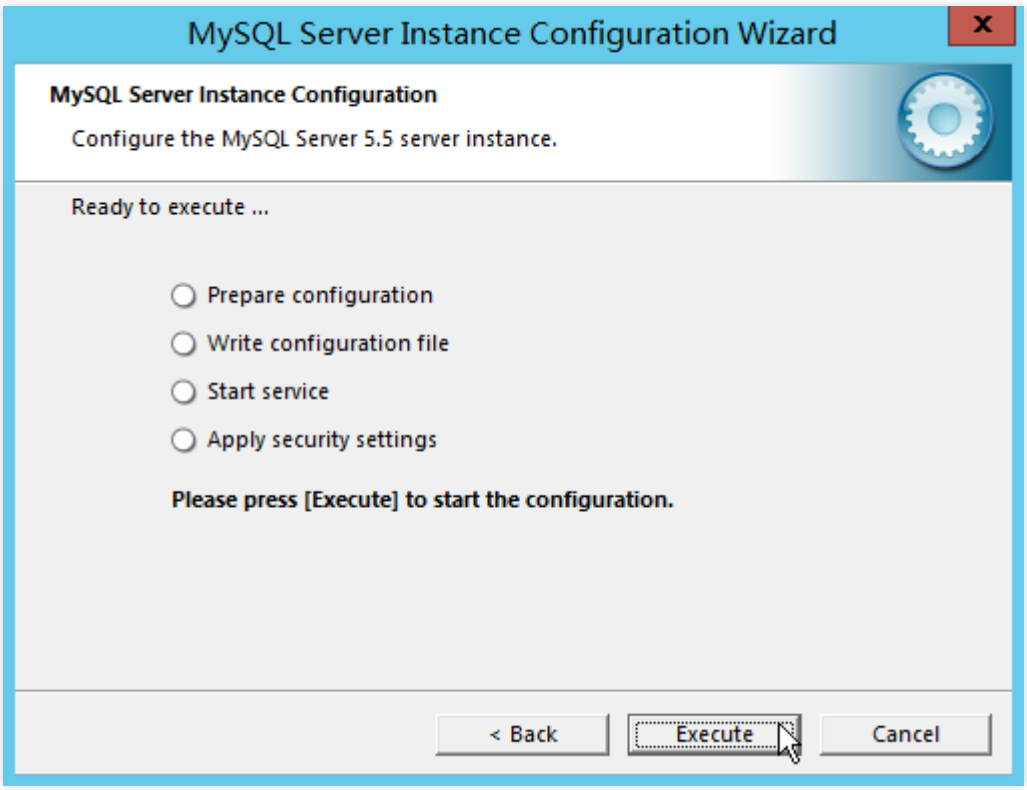
8. Configure the service options of MySQL. It is recommended to select both boxes to manage MySQL using command line.



9. Set the root password.



0. Complete the configuration. Click **Execute** to complete the installation.



Step 4: Login Test for MySQL

1. Click **Start** on the CVM and click the search icon. Enter `cmd` to open the administrator command box:

2. Enter the command `mysql -u root -p` and press **Enter**.

3. Log in to MySQL using the root password you set. The picture below indicates that MySQL has been installed and configured successfully.