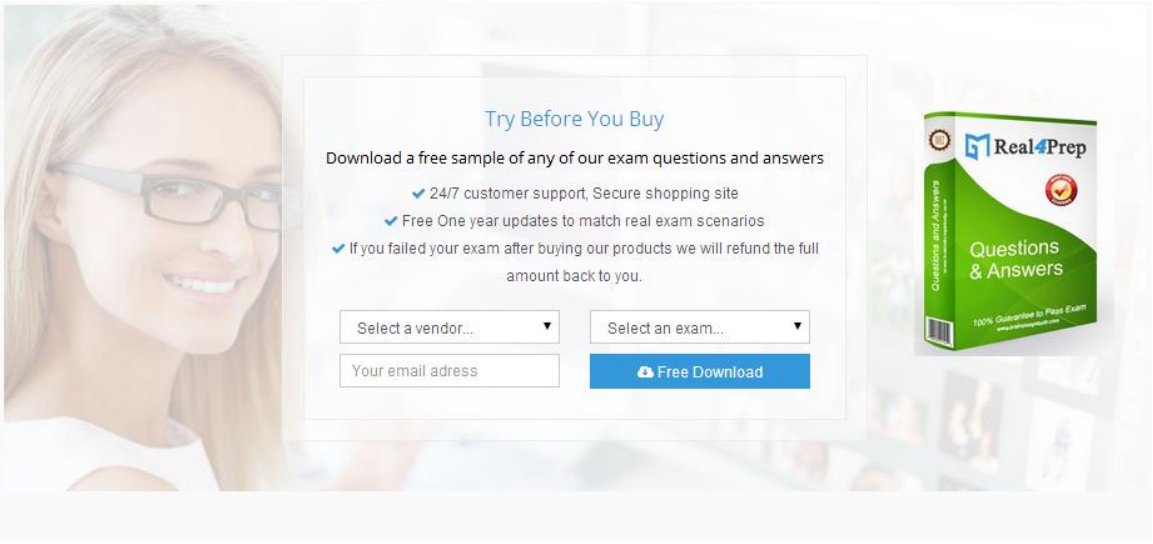
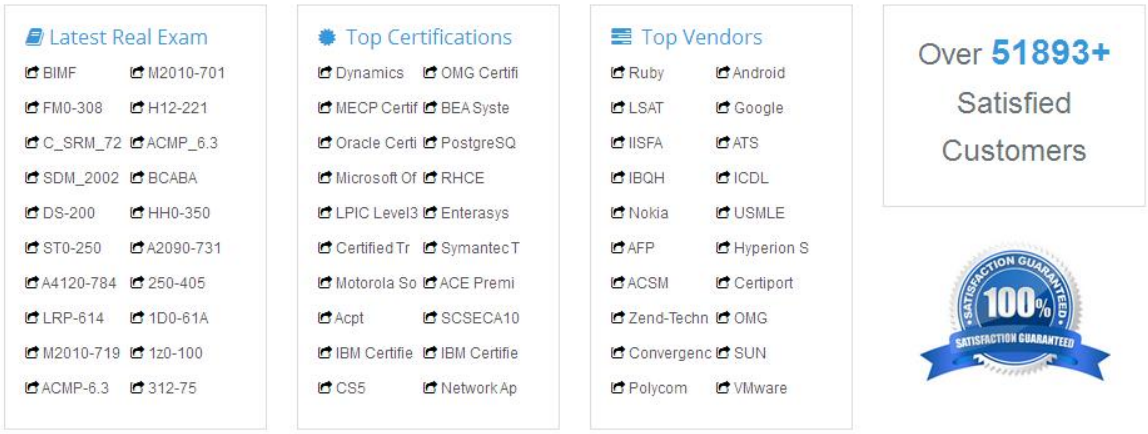
Real4Prep





http://www.real4prep.com

Latest Real Exam Prep Dumps for IT Exam Preparation

Exam : **101-500J**

Title : LPIC-1 Exam 101, Part 1 of 2, version 5.0 (101-500日本語版)

Vendor : Lpi

Version : DEMO

QUESTION NO: 1
BIOSはどこでブートローダーを検索しますか？
A.ブートデバイスの順序に関係なく、接続されているすべてのストレージメディア。
B.定義された起動デバイスの順序で接続されているすべてのストレージメディア。
C.定義された起動デバイスの順序のハードディスクドライブのみ。
D.最後に追加されたストレージメディアのみ。
E.BIOSは有効なブートローダーを検索する責任を負いません。
Answer: B
Explanation:
The BIOS (Basic Input/Output System) is a firmware program that is stored in a ROM chip on the motherboard, and it performs some basic tasks when the system is powered on, such as:
* initializing the hardware components and peripherals
* performing the power-on self-test (POST) to check the system integrity
* selecting a boot device from the boot order list
* loading and executing the bootloader program from the boot device
The boot device is the data carrier that contains the bootloader, which is a small program that is responsible for loading and executing the kernel. The boot device can be a hard disk, a USB drive, a CD-ROM, or a network device. The BIOS has a boot order list, which is a sequence of possible boot devices that the BIOS will search for a valid bootloader. The boot order list can be configured by the user through the BIOS setup utility, which is usually accessed by pressing a key such as F2, F10, or Del during the POST.
The BIOS will search for a bootloader by means of a special signature, which is a sequence of bytes that indicates the presence of a bootloader. The signature is usually located in the first sector of the boot device, which is called the boot sector or the master boot record (MBR). The BIOS will read the boot sector of each boot device in the boot order list, and check if the last two bytes are 0x55 and 0xAA, which are the standard boot signature. If the boot signature is found, the BIOS will load the boot sector into memory and execute it.
If the boot signature is not found, the BIOS will move on to the next boot device in the boot order list. If none of the boot devices have a valid boot signature, the BIOS will display an error message such as:
No bootable device - insert boot disk and press any key
References:
How does the BIOS know where the bootloader is located? - Super User
Bootloader: What is it and how does it work? - IONOS
Boot Process with Systemd in Linux - GeeksforGeeks
Linux Boot Process Step-by-Step Explained - javatpoint

QUESTION NO: 2
通常のファイルへのハードリンクを作成すると、エラーが返されます。この理由は何でしょうか？
A.ソースファイルは非表示です。
B.ソースファイルは読み取り専用です。
C.ソースファイルはシェルスクリプトです。
D.ソースファイルは既にハードリンクです。

E.ソースとターゲットは異なるファイルシステムにあります。
Answer: E
Explanation:
The most likely reason for getting an error when creating a hard link to an ordinary file is that the source and the target are on different filesystems. A hard link is a directory entry that points to the same inode as the original file. An inode is a data structure that stores the metadata and location of a file on a filesystem. A hard link cannot span across different filesystems because each filesystem has its own inode table and numbering scheme. Therefore, a hard link can only refer to a file that exists on the same filesystem as the link. This is mentioned in the article by How-To Geek1 and the man page of ln2. The other options are not valid reasons for getting an error when creating a hard link. The source file being hidden, read-only, or a shell script does not affect the ability to create a hard link, as long as the user has the appropriate permissions. The source file being already a hard link also does not prevent creating another hard link to the same file, as long as the maximum number of links per inode is not exceeded. This is explained in the article by Linuxize3 and the man page of ln

QUESTION NO: 3
次のコマンドのうち、SysV
initを実行しているシステムをメンテナンスタスクを実行しても安全な状態にするのはどれですか？（2つの正解を選択してください。）
A. shutdown -R 1 now
B. shutdown -single now
C. init 1
D. telinit 1
E. runlevel 1
Answer: C D
Explanation:
The commands init 1 and telinit 1 both bring a system running SysV init into a state in which it is safe to perform maintenance tasks. This state is also known as single-user mode or runlevel 1, where only the root user can log in and no network services are running. The command shutdown -R 1 now is incorrect, because it reboots the system instead of entering single-user mode. The command shutdown -single now is invalid, because the -single option does not exist for the shutdown command. The command runlevel 1 is also invalid, because runlevel is a command that displays the current and previous runlevels, not a command that changes the runlevel. References:
* 1: SysVinit - ArchWiki
* 2: Linux: How to write a System V init script to start, stop, and restart my own application or service - nixCraft
* 3: sysvinit - Gentoo wiki

QUESTION NO: 4
Linuxシステム内で使用されるinitシステムは次のうちどれですか？
（3つの正解を選択してください。）
A. startd

B. systemd
C. Upstart
D. SysInit
E. SysV init
Answer: B C E
Explanation:
systemd, Upstart, and SysV init are all init systems used within Linux systems. An init system is the first process executed by the kernel at boot time, which has a process ID (PID) of 1, and is responsible for starting and managing all other processes on the system. Different init systems have different features, advantages, and disadvantages. Some of the most common init systems are:
* systemd: A relatively new and modern init system that aims to provide a unified and efficient way of managing system and service states. It is compatible with SysV and LSB init scripts, and supports features such as parallel processing, socket activation, logging, job scheduling, and more. It is the default init system for many popular Linux distributions, such as Fedora, Debian, Ubuntu, Arch Linux, and others12.
* Upstart: An event-based init system developed by Ubuntu as a replacement for SysV init. It starts and stops system tasks and processes based on events, such as hardware changes, network availability, filesystem mounting, etc. It is a hybrid init system that uses both SysV and systemd scripts, and supports features such as parallel processing, dependency tracking, logging, and more. It is the default init system for some older versions of Ubuntu, and some other Linux distributions, such as Linux Mint and Chrome OS12.
* SysV init: A mature and traditional init system that follows the System V (SysV) design of Unix operating systems. It uses a series of runlevels to define the state of the system, and executes scripts in the /etc/rc.d or /etc/init.d directories according to the current runlevel. It is simple and stable, but lacks some features of modern init systems, such as parallel processing, event handling, dependency tracking, etc. It is still used by some Linux distributions, such as Slackware, Gentoo, and others12.
1: 6 Best Modern Linux 'init' Systems (1992-2023) - Tecmint 2: 10 Best Linux init systems as of 2023 - Slant.

QUESTION NO: 5
gzipで圧縮されたtarアーカイブの内容を表示するコマンドは次のうちどれですか？
A. gzip archive.tgz | tar xvf -
B. tar ztf archive.tgz
C. gzip -d archive.tgz | tar tvf -
D. tar cf archive.tgz
Answer: B
Explanation:
The command that displays the contents of a gzip compressed tar archive is tar ztf archive.tgz. This command uses the following options:
-z: Tells tar to read or write archives through gzip, allowing it to work on compressed files directly. -t: Lists the contents of an archive without extracting it. -f archive.tgz: Specifies the name of the archive file.

The output of this command will show the names of the files and directories stored in the archive, one per line. For example, if the archive contains three files named file1, file2, and file3, the output will be:
file1 file2 file3
The other commands are incorrect for the following reasons:
* gzip archive.tgz | tar xvf -: This command will decompress the archive using gzip and pipe it to tar, which will extract the files to the current directory. The - option tells tar to read the archive from the standard input. This command does not display the contents of the archive, but rather extracts them.
* gzip -d archive.tgz | tar tvf -: This command is similar to the previous one, except that it uses the -d option for gzip to decompress the archive instead of compressing it, and the -t option for tar to list the contents instead of extracting them. However, this command is redundant and inefficient, as tar can handle compressed archives directly with the -z option. Also, the -d option for gzip will delete the original archive file after decompression, which may not be desirable.
* tar cf archive.tgz: This command will create a new archive named archive.tgz from the files and directories given as arguments. However, this command does not use the -z option, so the archive will not be compressed with gzip. Also, this command does not display the contents of the archive, but rather creates it.

QUESTION NO: 6
Windows NTでセーフモードを起動するには、どのファンクションキーを使用しますか？
A.F10
B.F8
C.F6
D.Windows NTはセーフモードをサポートしていません
Answer: D
Explanation:
The function key that is used to start Safe Mode in Windows NT is none of the above, because Windows NT does not support Safe Mode. Safe Mode is a diagnostic mode of Windows that starts the system with minimal drivers and services, allowing the user to troubleshoot problems and restore the system to a normal state1. Safe Mode was introduced in Windows 95 and later versions, but not in Windows NT 4.0 and earlier2.
The other options are incorrect for the following reasons:
* F10. This function key is used to access the Recovery Console in Windows XP, which is a command- line interface that allows the user to perform various administrative tasks, such as repairing the boot sector, restoring the registry, or copying files3. The Recovery Console is not the same as Safe Mode, and it is not available in Windows NT.
* F8. This function key is used to access the Advanced Boot Options menu in Windows Vista and later versions, which allows the user to choose from various boot modes, including Safe Mode, Last Known Good Configuration, Debugging Mode, and others4. In Windows NT, pressing F8 during startup only displays a simple menu with three options: Normal, VGA mode, and Boot Logging5. None of these options are equivalent to Safe Mode.
* F6. This function key is used to load additional drivers during the installation of Windows, such as SCSI or RAID drivers, from a floppy disk or a USB flash drive6. This function key has

nothing to do with Safe Mode, and it is not relevant after the installation is completed.
:
Start your PC in safe mode in Windows - Microsoft Support
How do you boot NT into safe mode? | TechRepublic
How to install the Recovery Console to your hard disk
Advanced startup options (including safe mode) - Windows Help
How to Boot Windows NT in Safe Mode
How to install third-party SCSI or RAID driver

QUESTION NO: 7
ログイン試行に使用される秘密鍵の場所を指定する ssh
コマンドのパラメータはどれですか?
(値やパラメータを指定せずにオプション名のみを指定します)
Answer:
ssh-key

QUESTION NO: 8
SysV initを使用するときにシステムを再起動するコマンドは次のうちどれですか？
（2つの正解を選択してください。）
A. shutdown -r now
B. shutdown -r "rebooting"
C. telinit 6
D. telinit 0
E. shutdown -k now "rebooting"
Answer: A C
Explanation:
The shutdown command is used to bring the system down in a safe and controlled way. It can take various options and arguments, such as the time of shutdown, the message to broadcast to users, the halt or reboot mode, etc. The option -r instructs the shutdown command to reboot the system after shutting down. The argument now means to shut down immediately. Therefore, shutdown -r now will reboot the system without delay. The telinit command is used to change the run level of the system. It takes a single argument that specifies the new run level. The run level 6 is reserved for rebooting the system. Therefore, telinit 6 will also reboot the system. The other options are either incorrect or irrelevant. shutdown -r "rebooting" will also reboot the system, but with a delay of one minute and a message to the users. telinit 0 will halt the system, not reboot it. shutdown -k now "rebooting" will only send a warning message to the users, but not actually shut down or reboot the system. References: LPI Linux Essentials - 1.101.2, LPI Linux Administrator - 101.3

QUESTION NO: 9
次のコマンドのうち、lsの出力を標準エラーにリダイレクトするのはどれですか？
A. ls >-1
B. ls <<ERR
C. ls >&2
D. ls >>2

E. ls |error

Answer: C

Explanation:

This command redirects the output of ls to standard error, which is file descriptor 2 by default. The syntax of the command is:

ls >& file_descriptor

The ls command is a utility that lists the files and directories in the current working directory or a specified directory. The >& symbol is a redirection operator that redirects the standard output of a command to another file descriptor, which can be a file, a device, or another stream. The file_descriptor is the number of the file descriptor to which the output is redirected.

Therefore, the command ls >&2 will run the ls command and redirect its output to file descriptor 2, which is standard error. This means that the output of ls will not be displayed on the screen, but sent to the standard error stream, which can be used for error handling or logging purposes.

The other commands are incorrect for the following reasons:

* A, ls >-1: This command will not redirect the output of ls to standard error, but it will cause an error.

The > symbol is a redirection operator that redirects the standard output of a command to a file or device, overwriting any existing content. The -1 argument is not a valid file name or device name, and it will cause the shell to report an error and exit.

* B, ls <<ERR: This command will not redirect the output of ls to standard error, but it will use a here document as the standard input of ls. The << symbol is a redirection operator that redirects the standard input of a command from a here document, which is a block of text that follows the command. The ERR argument is the delimiter that marks the end of the here document. However, the ls command does not read from the standard input, and it will ignore the here document. The command will display the files and directories in the current working directory on the screen, as usual.

* D, ls >>2: This command will not redirect the output of ls to standard error, but it will append the output of ls to a file named 2. The >> symbol is a redirection operator that redirects the standard output of a command to a file or device, appending to any existing content. The 2 argument is the name of the file to which the output is appended. If the file does not exist, it will be created. The command will not display anything on the screen, but write the output of ls to the file 2.

* E, ls |error: This command will not redirect the output of ls to standard error, but it will pipe the output of ls to another command named error. The | symbol is a pipe operator that redirects the standard output of one command to the standard input of another command. The error argument is the name of the command that receives the output of ls as its input. However, there is no such command named error in the Linux system, and the shell will report an error and exit.

QUESTION NO: 10

通常のブートシーケンス中に出力されたカーネルからのメッセージを表示するコマンドはどれですか？

Answer:

dmesg
Explanation:
The command dmesg will display messages from the kernel that were output during the normal boot sequence. The dmesg command reads the kernel ring buffer, which is a data structure that stores the most recent messages generated by the kernel. The dmesg command can also be used to display messages from the kernel that were output after the boot sequence, such as hardware events, driver messages, or system errors.
The dmesg command has various options to filter, format, or save the output. For example, dmesg -T will display human-readable timestamps for each message, and dmesg -w will display the messages in real time as they occur. References:
* 1: How to view all boot messages in Linux after booting? - Super User
* 2: dmesg(1) - Linux manual page
* 3: Kernel ring buffer - Wikipedia

QUESTION NO: 11
次のコマンドについて正しいのは何ですか?
Nmcli デバイスの Wi-Fi 接続 WIFIoI
A. NetworkManagerはSSID wifioiで新しいパブリックホットスポットを開きます
B. NetworkManager は、wifioi という名前の未構成の新しい仮想ネットワーク
インターフェイスを作成します。
C. NetworkManagerは新しいWi-Fi接続wifioiを作成し、それをアクティブ化します。
D. Wi-Fi接続が存在しない場合、NetworkManagerはエラーを返します。
E. wifioi は無効な Wi-Fi デバイスであるため、NetworkManager はエラーを報告します
Answer: C

QUESTION NO: 12
/ etc /
passwdから口グインシェルと共にユーザー名のリストを生成するコマンドは何ですか？
A. column -s : 1,7 /etc/passwd
B. chop -c 1,7 /etc/passwd
C. colrm 1,7 /etc/passwd
D. cut -d: -f1,7 /etc/passwd
Answer: D
Explanation:
The cut command is used to extract sections from each line of a file or input stream. The -d option specifies the delimiter that separates the fields in each line. The -f option specifies the fields to select. In this case, the delimiter is a colon (:) and the fields are 1 and 7. The first field is the user name and the seventh field is the login shell. Therefore, the cut command with these options will generate a list of user names and their login shells from /etc/passwd. References:
* Linux cut Command Explained with 6 Examples
* How to Use the Linux cut Command
* cut command in Linux with examples

QUESTION NO: 13

teeコマンドのどのオプションを使用すると、既存のファイルの内容を上書きする代わりに、出力ファイルの最後に出力が連結されますか？

A.-a

B.-c

C.--no-clobber

D.--continue

Answer: A

Explanation:

The -a option to the tee command will cause the output to be appended to the end of the output file instead of overwriting the existing file contents. The tee command reads from standard input (STDIN) and writes to standard output (STDOUT) and one or more files simultaneously. For example, ls | tee file.txt will display the output of the ls command and also write it to file.txt. If file.txt already exists, it will be overwritten unless the - a option is used. References: LPI Exam 101 Detailed Objectives, Topic 103: GNU and Unix Commands, Weight: 25, Objective 103.3: Perform basic file management, tee command

QUESTION NO: 14

次のタスクのうち、XDM や KDM などのディスプレイ
マネージャーによって処理されるのはどれですか。(正しい回答を 2 つ選択してください。)

A. 新しいモニターやプロジェクターなどの追加デバイスが接続されたときに設定します

B. ユーザーのデスクトップ環境を起動して準備します

C. 現在のグラフィックデバイスとモニターのX11構成ファイルを作成します。

D. ユーザーが一定時間非アクティブだった場合に画面をロックします

E. ユーザーのログインを処理する

Answer: BE

QUESTION NO: 15

シャドウ パスワードは、標準の非シャドウ
パスワードと比較して、パスワードのセキュリティをどのように向上させるのでしょうか?

A. 通常のユーザーはシャドウパスワードのパスワードハッシュにアクセスできません

B. すべてのシャドウパスワードは45日間有効で、その後は変更する必要があります。

C. システムのホスト キーは、すべてのシャドウ パスワードの暗号化に使用されます。

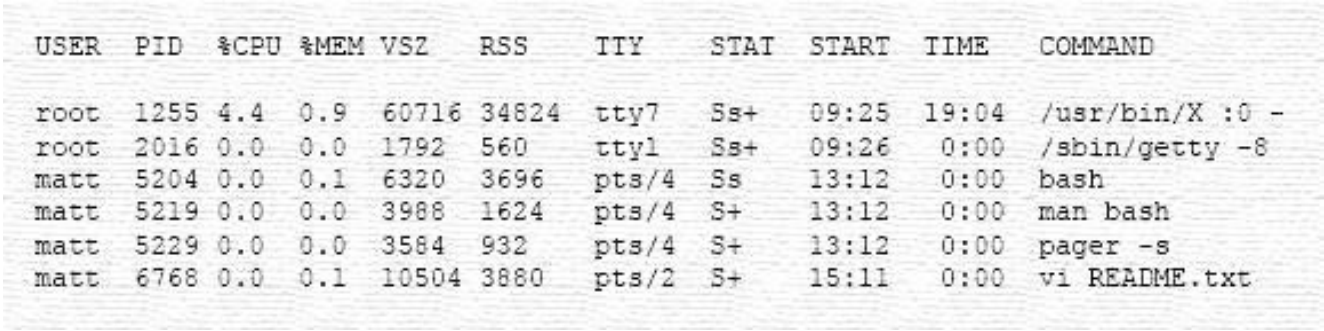
D. シャドウパスワードは常にユーザーの秘密鍵と一致する公開鍵と組み合わせられます。

E.
シャドウパスワードはプレーンテキストで保存され、弱いパスワードかどうかチェックできます。

Answer: A

QUESTION NO: 16

次のコマンドのうち、次の出力を生成するのはどれですか？



```
USER  PID  %CPU %MEM VSZ   RSS   TTY    STAT  START  TIME   COMMAND

root  1255 4.4  0.9  60716 34824 tty7   Ss+   09:25  19:04  /usr/bin/X :0 -
root  2016 0.0  0.0  1792  560   tty1   Ss+   09:26   0:00  /sbin/getty -8
matt  5204 0.0  0.1  6320  3696  pts/4  Ss    13:12   0:00  bash
matt  5219 0.0  0.0  3988  1624  pts/4  S+    13:12   0:00  man bash
matt  5229 0.0  0.0  3584  932   pts/4  S+    13:12   0:00  pager -s
matt  6768 0.0  0.1  10504 3880  pts/2  S+    15:11   0:00  vi README.txt
```

A. jobs

B. proclist

C. netstat

D. ps

Answer: D

Explanation:

The ps command will produce the output shown in the image. The ps command displays information about the processes running on the system. The output format can be customized by using different options. For example, ps -aux will show all processes with detailed information, such as user, PID, CPU, memory, command, and so on. The output in the image matches the format of ps -aux. The jobs command will show the status of jobs in the current shell. The proclist command is not a valid Linux command. The netstat command will show network connections, routing tables, and statistics. References: LPI Exam 101 Detailed Objectives, Topic 103: GNU and Unix Commands, Weight: 25, Objective 103.3: Perform basic file management, ps command, jobs command, netstat command

QUESTION NO: 17

次のコマンドのうち、fooコマンドの出力を画面に表示し、/ tmp /
foodataというファイルに書き込むコマンドはどれですか？

A. foo | less /tmp/foodata

B. foo | cp /tmp/foodata

C. foo > /tmp/foodata

D. foo | tee /tmp/foodata

E. foo > stdout >> /tmp/foodata

Answer: D

Explanation:

This command will display the output of the foo command on the screen and also write it to a file called /tmp
/foodata. The syntax of the command is:
foo | tee [options] [file]
The foo command is any command that produces some output. The | symbol is a pipe operator that redirects the standard output of one command to the standard input of another command. The tee command reads from the standard input and writes to both the standard output and one or more files. The options can modify the behavior of the tee command, such as appending to the file instead of overwriting it, or ignoring interrupt signals. The file is the name of the file to which the output is written. If no file is given, the tee command will only

write to the standard output.
Therefore, the command foo | tee /tmp/foodata will run the foo command, pipe its output to the tee command, which will display the output on the screen and write it to the file /tmp/foodata.
The other commands are incorrect for the following reasons:
* A, foo | less /tmp/foodata: This command will not write the output of the foo command to a file, but it will display the output of the foo command on the screen in a pager. The less command is a program that allows the user to view and scroll through a file or the output of a command. The syntax of the command is:
foo | less [options] [file]
The foo command is any command that produces some output. The | symbol is a pipe operator that redirects the standard output of one command to the standard input of another command. The less command reads from the standard input or a file and displays it on the screen in a pager. The options can modify the behavior of the less command, such as setting the number of lines per screen, or searching for a pattern. The file is the name of the file to be viewed. If no file is given, the less command will read from the standard input.
Therefore, the command foo | less /tmp/foodata will run the foo command, pipe its output to the less command, which will display the output on the screen in a pager. However, the /tmp/foodata argument will be ignored by the less command, because it will read from the standard input instead of the file. The command will not write anything to the file /tmp/foodata.
* B, foo | cp /tmp/foodata: This command will not work as expected, because it has several errors. First, the cp command is not a valid command to write the output of a command to a file. The cp command is used to copy files or directories from one location to another. The syntax of the command is:
cp [options] source destination
The options can modify the behavior of the cp command, such as preserving the attributes of the files, or creating backups of the existing files. The source is the name of the file or directory to be copied. The destination is the name of the file or directory where the source is copied to.
Second, the pipe operator is not a valid way to redirect the output of a command to the cp command. The pipe operator redirects the standard output of one command to the standard input of another command. However, the cp command does not read from the standard input, but from the source argument. Therefore, the command foo | cp /tmp/foodata will run the foo command, pipe its output to the cp command, which will ignore the standard input and report an error for missing the destination argument. The command will not write anything to the file /tmp/foodata.
* C, foo > /tmp/foodata: This command will not display the output of the foo command on the screen, but it will write it to a file called /tmp/foodata. The > symbol is a redirection operator that redirects the standard output of a command to a file or device, overwriting any existing content. The syntax of the command is:
foo > file
The foo command is any command that produces some output. The > symbol redirects the standard output of the foo command to the file. The file is the name of the file to which the output is written.

Therefore, the command foo > /tmp/foodata will run the foo command, redirect its output to the file /tmp
/foodata, and overwrite any previous content. The command will not display anything on the screen.
* E, foo > stdout >> /tmp/foodata: This command will not work as expected, because it has several errors.
First, the stdout argument is not a valid file name or device name. The stdout is an abbreviation for the standard output, which is a stream that a program uses to write its output. However, the stdout is not a file or device that can be used as a destination for the redirection operator. Second, the >> symbol is a redirection operator that redirects the standard output of a command to a file or device, appending to any existing content. The syntax of the command is:
foo >> file
The foo command is any command that produces some output. The >> symbol redirects the standard output of the foo command to the file. The file is the name of the file to which the output is appended.
Therefore, the command foo > stdout >> /tmp/foodata will run the foo command, redirect its output to the stdout argument, which will cause an error, and then redirect its output again to the file /tmp/foodata, which will append the output to the file. The command will not display anything on the screen.
:
Linux Tee Command with Examples | Linuxize
tee command in Linux with examples - GeeksforGeeks
Linux tee command explained for beginners (6 examples) - HowtoForge
Command Options and Examples of Tee Command in Linux - UbuntuPIT
Linux tee Command Explained for Beginners (6 Examples) - Linux Handbook.

QUESTION NO: 18
次のコマンドのうち、連続するすべてのスペースを1つのスペースに減らすものはどれですか？
A. tr '\s' ' ' < a.txt > b.txt
B. tr -c ' ' < a.txt > b.txt
C. tr -d ' ' < a.txt > b.txt
D. tr -r ' ' '\n' < a.txt > b.txt
E. tr -s ' ' < a.txt > b.txt
Answer: E
Explanation:
The command that will reduce all consecutive spaces down to a single space is tr -s ' ' < a.txt > b.txt. This command uses the following options and syntax:
* -s: Squeezes repeated characters listed in the first set with single occurrence.
* ' ': Specifies a space character as the first set.
* < a.txt: Redirects the input from a file named a.txt.
* b.txt: Redirects the output to a file named b.txt.
The output of this command will be a new file called b.txt that contains the same text as a.txt, except that any sequence of multiple spaces will be replaced by a single space. For example,

if the file a.txt contains the following text:
This is a text file with multiple spaces.
The file b.txt will contain the following text:
This is a text file with multiple spaces.
The other commands are incorrect for the following reasons:
* A. tr '\s' ' ' < a.txt > b.txt: This command will replace every whitespace character (\s) with a space character, which will not reduce the number of spaces, but rather convert tabs and newlines into spaces.
* B. tr -c ' ' < a.txt > b.txt: This command will complement the first set, meaning that it will apply the operation to all characters that are not spaces. This will not affect the spaces at all, but rather squeeze all other characters.
* C. tr -d ' ' < a.txt > b.txt: This command will delete all spaces from the input, which will not reduce them to a single space, but rather remove them completely.
* D. tr -r ' ' '\n' < a.txt > b.txt: This command will replace all spaces with newlines, which will not reduce the spaces, but rather create a new line for each word.
References:
[LPI Exam 101 Detailed Objectives], Topic 103: GNU and Unix Commands, Objective 103.2: Process text streams using filters, Weight: 3, Key Knowledge Areas: Use of tr to translate characters or to squeeze and/or delete them.
Tr Command in Linux with Examples, Topic: How to squeeze a sequence of repetitive characters using -s option.

QUESTION NO: 19
次のコマンドのうち、/ dev /
sda1上のext3ファイルシステムが起動中に完全なファイルシステムチェックを実行するまでの日数を変更するのはどれですか？
A. tune2fs -d 200 /dev/sda1
B. tune2fs -c 200 /dev/sda1
C. tune2fs -i 200 /dev/sda1
D. tune2fs -n 200 /dev/sda1
E. tune2fs --days 200 /dev/sda1
Answer: C
Explanation:
The correct command to change the number of days before the ext3 filesystem on /dev/sda1 has to run through a full filesystem check while booting is tune2fs -i 200 /dev/sda1. This command sets the interval between two filesystem checks to 200 days. The tune2fs command allows you to view and change various filesystem parameters on Linux ext2, ext3, or ext4 filesystems. The -i option specifies the time interval between checks. The other options are incorrect because they use the wrong parameters for the tune2fs command. Option A is wrong because the -d option is not supported by the tune2fs command. Option B is wrong because the -c option sets the maximum number of mounts, not days, before a check. Option D is wrong because the -n option is not supported by the tune2fs command. Option E is wrong because the --days option is not supported by the tune2fs command.
References:
* [LPI Linux Essentials - 2.2 Mounting, Unmounting Filesystems]

* 15 tune2fs command examples in Linux [Cheat Sheet] - GoLinuxCloud
* Linux tune2fs command With Examples - GeeksforGeeks
* tune2fs command-file system management - Linuxstar
* tune2fs Command Examples - Gianforte School of Computing

QUESTION NO: 20
Bashシェルから、次のコマンドのどれがファイルから命令を直接実行しますか
サブシェルを起動せずに/usr/local/bin/runme.sh？（2つの回答を選択してください。）
A. source /usr/local/bin/runme.sh
B. . /usr/local/bin/runme.sh
C. /bin/bash /usr/local/bin/runme.sh
D. /usr/local/bin/runme.sh
E. run /usr/local/bin/runme.sh
Answer: A B
Explanation:
The commands that directly execute the instruction from the file /usr/local/bin/runme.sh without starting a subshell are source /usr/local/bin/runme.sh and . /usr/local/bin/runme.sh. These commands use the source or dot builtins, which read and execute commands from the given file in the current shell environment. This means that any changes made by the file, such as setting variables, defining functions, or changing directories, will affect the current shell. This is different from running the file as a script, which will create a new shell process and execute the commands in a separate environment. The source or dot commands are useful for loading configuration files, such as ~/.bashrc or /etc/profile, or for running scripts that modify the current state of the shell.
The other commands are incorrect for the following reasons:
* C. /bin/bash /usr/local/bin/runme.sh: This command will run the file as a script using the /bin/bash interpreter. This will create a new shell process and execute the commands in a separate environment.
Any changes made by the file will not affect the current shell.
* D. /usr/local/bin/runme.sh: This command will also run the file as a script, but using the interpreter specified by the shebang line (#!) at the beginning of the file. If the file does not have a shebang line, it will use the default shell interpreter, which may or may not be /bin/bash. This will also create a new shell process and execute the commands in a separate environment. Any changes made by the file will not affect the current shell.
* E. run /usr/local/bin/runme.sh: This command is not valid, as there is no builtin or external command called run. This will produce an error message.
References:
[LPI Exam 101 Detailed Objectives], Topic 103: GNU and Unix Commands, Objective 103.1: Work on the command line, Weight: 4, Key Knowledge Areas: Use of source and ..
[Bash Reference Manual], Section 4.2: Bash Builtin Commands, Subsection 4.2.5: Bourne Shell Builtins.

QUESTION NO: 21
次のコマンドのうち、特定のユーザーのクォータを変更するのはどれですか？
A. edquota

B. repquota
C. quota -e
D. quota
Answer: A
Explanation:
The correct command to change the quota for a specific user is edquota. This command allows you to edit the quota limits for a user, a group, or a file set. You can specify the name of the user that you want to edit the quotas for after the command. For example, to change the disk quota for user 'linuxconfig', you can use the following command:
sudo edquota -u linuxconfig
This command will open an editor with the current quota information for the user 'linuxconfig'. You can modify the soft and hard limits for the block and inode usage as per your requirements. You can also use the - p option to copy the quota settings from another user. For example, to copy the quota settings from user
'ramesh' to user 'linuxconfig', you can use the following command:
sudo edquota -p ramesh -u linuxconfig
The other commands are not suitable for changing the quota for a specific user. The repquota command displays a summary of the current quota usage and limits for the users or groups. The quota -e command turns off the disk quota for the current user. The quota command shows the disk quota and usage for the current user or for the users specified on the command line. For more information on how to use disk quota on Linux, you can refer to the following articles:
* How to use disk quota on Linux with examples
* 5 Steps to Setup User and Group Disk Quota on UNIX / Linux

QUESTION NO: 22
コマンドfooが呼び出されたときに実行される実行可能ファイルへのパスを表示するコマンドは次のうちどれですか？
A. lsattr foo
B. apropos foo
C. locate foo
D. whatis foo
E. which foo
Answer: E
Explanation:
This command will display the path to the executable file that would be executed when the command foo is invoked. The syntax of the command is:
which [options] command
The which command is a utility that searches the directories listed in the PATH environment variable for the executable file that matches the given command. The options can modify the behavior of the which command, such as displaying all matches, ignoring aliases, or showing the version. The command is the name of the command to be located.
Therefore, the command which foo will search the PATH directories for the executable file named foo and print its full path on the standard output. If there are multiple matches, the

command will print the first one found. If there is no match, the command will print nothing and return an exit status of 1.
The other commands are incorrect for the following reasons:
* A, lsattr foo: This command will not display the path to the executable file, but it will display the file attributes of the file named foo in the current directory. The syntax of the command is:
lsattr [options] [file]
The lsattr command is a utility that lists the file attributes on a Linux second extended file system. The options can modify the behavior of the lsattr command, such as displaying the output in long format, recursing into subdirectories, or suppressing errors. The file is the name of the file whose attributes are to be listed. If no file is given, the command will list the attributes of all files in the current directory.
Therefore, the command lsattr foo will list the file attributes of the file named foo in the current directory, if it exists. If it does not exist, the command will report an error and return an exit status of 1.
* B, apropos foo: This command will not display the path to the executable file, but it will display a list of manual page names and descriptions that contain the keyword foo. The syntax of the command is:
apropos [options] keyword
The apropos command is a utility that searches the whatis database for the keyword and prints the manual page names and descriptions that match. The whatis database is a set of files containing short descriptions of system commands and programs. The options can modify the behavior of the apropos command, such as using regular expressions, ignoring case, or displaying the section numbers. The keyword is the word to be searched in the whatis database.
Therefore, the command apropos foo will search the whatis database for the word foo and print the manual page names and descriptions that contain it. If there are no matches, the command will print nothing and return an exit status of 1.
* C, locate foo: This command will not display the path to the executable file, but it will display a list of file names that contain the string foo. The syntax of the command is:
locate [options] pattern
The locate command is a utility that searches a database of file names and prints the file names that match the given pattern. The database is updated periodically by the updatedb command and may not reflect the current state of the file system. The options can modify the behavior of the locate command, such as using regular expressions, ignoring case, or limiting the number of results. The pattern is the string to be matched in the file names.
Therefore, the command locate foo will search the database of file names and print the file names that contain the string foo. If there are no matches, the command will print nothing and return an exit status of 1.
* D, whatis foo: This command will not display the path to the executable file, but it will display a short description of the command or program named foo. The syntax of the command is:
whatis [options] name
The whatis command is a utility that searches the whatis database for the name and prints the manual page name and description that match. The whatis database is a set of files containing short descriptions of system commands and programs. The options can modify

the behavior of the whatis command, such as displaying the section numbers, using wildcards, or searching in a specific section. The name is the name of the command or program to be described.
Therefore, the command whatis foo will search the whatis database for the name foo and print the manual page name and description that match. If there are no matches, the command will print nothing and return an exit status of 1.
:
Which Command in Linux [Explained with Examples]
How to Use the which Command on Linux - How-To Geek
which command in Linux with examples - GeeksforGeeks
How to Use the which Command in Linux - phoenixNAP

QUESTION NO: 23
/proc 内のどのファイルが、さまざまなカーネル ドライバーによって使用される IRQ を記述しますか? (パスなしでファイル名のみを指定します。)
Answer:
中断

QUESTION NO: 24
次のルーティング テーブルがあるとします。

```
Kernel IP routing table
Destination     Gateway         Genmask         Flags Metric Ref    Use Iface
0.0.0.0         192.168.178.1   0.0.0.0         UG    0      0        0 wlan0
192.168.1.0     0.0.0.0         255.255.255.0   U     0      0        0 eth0
192.168.2.0     192.168.1.1     255.255.255.0   U     0      0        0 eth0
192.168.178.0   0.0.0.0         255.255.255.0   U     9      0        0 wlan0
```

宛先 192 168 2 150 への送信パケットはどのように処理されるでしょうか?
A. デフォルト ルータ 192 168 178 1 onwlan0 に渡されます。
B. デバイス eth0 に直接送信されます
C. eth0のデフォルトルータ255 255 255 0に渡されます
D. ルータ192 168.1.1 oneth0に渡されます
E. デバイス wlan0 上で直接送信されます。
Answer: D

QUESTION NO: 25
Debianシステムで利用可能なすべてのパッケージで全文検索を実行するために使用できるコマンドは次のうちどれですか？
A.apt
B.apt-cache
C.apt-get
D.apt-search
E.dpkg
Answer: B
Explanation:

The command apt-cache can be used to perform a full text search on all available packages on a Debian system. It searches the package names and the descriptions for an occurrence of the regular expression given as a keyword and prints out the package name and the short description1. The syntax is: apt-cache search keyword. For example, apt-cache search openssh will return a list of packages related to OpenSSH2. The other commands are not suitable for this task because:
* apt is a high-level command-line tool that provides a user-friendly way to manage packages, but it does not have a search option3.
* apt-get is a low-level command-line tool that handles the installation and removal of packages, but it does not have a search option4.
* apt-search is not a valid command.
* dpkg is a tool to install, build, remove and manage Debian packages, but it does not have a search option5. It can only list the installed packages with the option -l4. References:
* How To Search For Available Packages From Command Line In Debian, Ubuntu Or Linux Mint [APT]
- Linux Uprising Blog
* apt(8) - apt - Debian buster - Debian Manpages
* How to List Installed Packages on Debian | Linuxize
* Debian / Ubuntu Linux search package names with apt-cache command
* dpkg(1) - dpkg - Debian buster - Debian Manpages

QUESTION NO: 26
yum コマンドのどのオプションがシステム全体を更新しますか?
(追加のパラメータなしでオプション名のみを指定します。)
A. 更新
B. アップグレード
C. 更新/アップグレード
Answer: A
Explanation:
yumコマンドは、Red Hat Enterprise
Linuxやその他のRPMベースのシステム上のソフトウェアパッケージを管理するためのツールです。yum
updateオプションは、インストールされているパッケージのバージョンを確認し、リポジトリから利用可能な最新バージョンをインストールすることで、システム全体をアップデートします。yum
upgradeオプションも同様の機能に加え、システムに不要になった古いパッケージを削除します。どちらのオプションも、アップデートまたはアップグレードのプロセスを進める前に、ユーザーに確認を求めます。参考資料：
* Red Hat Enterprise Linux 用 Yum コマンド チートシート
* 「yum update」を初心者向けに解説！
* CentOS 7にアップデートをインストールする方法
* yum コマンドのどのオプションでシステム全体が更新されますか?

QUESTION NO: 27
次のカーネルパラメーターのどれがカーネルにほとんどのブートメッセージを抑制するよう

に指示しますか？
A. silent
B. verbose=0
C. nomesg
D. quiet
Answer: D
Explanation:
The quiet kernel parameter instructs the kernel to suppress most boot messages, except for critical errors12. The quiet parameter can be added to the GRUB_CMDLINE_LINUX_DEFAULT variable in the /etc /default/grub file and then run sudo update-grub to apply the changes3. The quiet parameter can also be used in combination with other parameters, such as splash, to enable a graphical boot screen4.
The other options in the question are not valid or do not have the same functionality as the quiet parameter:
* silent: There is no such kernel parameter in Linux.
* verbose=0: This parameter is used to set the verbosity level of the kernel messages, but it does not suppress them completely. The valid values for this parameter are from 0 (quiet) to 7 (debug)5.
* nomesg: This parameter is used to disable all kernel messages on the console, including the emergency ones. This parameter is not recommended for normal use, as it can hide critical errors and prevent troubleshooting.
:
1: Getting the Kernel Command-Line Parameters | Baeldung on Linux 2: How to mute kernel messages at startup in Arch Linux? 3: boot - How to turn off the filesystem check message which occures while booting - Ask Ubuntu 4: [How to enable a graphical boot screen on Ubuntu 18.04 LTS - LinuxConfig.org] 5: [Kernel parameters - ArchWiki] : [Linux Kerne l Parameters - SysTutorials]

QUESTION NO: 28
パッケージが初めてインストールされたかのように、どのDebianパッケージ管理ツールが、既にインストールされている特定のパッケージの設定に関する質問をしますか？
（パスまたはパラメーターなしでコマンドのみを指定します。）
Answer:
dpkg-reconfigure
Explanation:
The command dpkg-reconfigure is a Debian package management tool that asks the configuration questions for a specific already installed package just as if the package were being installed for the first time. It can be used to reconfigure a package that was previously installed with default settings, or to change the settings of a package that requires user input during installation. It can also be used to fix a broken configuration file or to restore the original configuration file of a package. References:
* Debian Reference: Chapter 2. Debian package management
* LPI Linux Essentials: 1.3 Package Management

QUESTION NO: 29
特定のホストにサービスへのアクセスを許可するように TCP
ラッパー構成を編集した後、これらの変更はいつ有効になりますか?
A. 新しい設定は、それぞれのサービスを再起動した後に有効になります。
B. 新しい設定は次回のシステム再起動時に有効になります
C. 新しい構成は、サービスへの最後の接続が閉じられたときに有効になります。
D. 新しい設定はtcpdサービスの再起動後に有効になります。
E. 新しい設定は、すべての新しい接続に対して直ちに有効になります。
Answer: E

QUESTION NO: 30
xargsコマンドの目的は何ですか？
A.引数をXサーバーに渡します。
B.標準入力（STDIN）を読み取り、実行するコマンドラインを作成します。
C.シェルスクリプトが可変引数リストを取るのに役立ちます。
D.グラフィカルに質問し、シェルに回答を返します。
E.通常、短いオプションのみを受け入れるコマンドに長いオプションを指定できます。
Answer: B
Explanation:
The purpose of the xargs command is to read standard input (STDIN) and build up command lines to execute.
The xargs command can be used to pass arguments to another command that does not accept input from a pipe. For example, rm | xargs echo will echo the arguments passed to the rm command. The xargs command can also limit the number of arguments per command line, insert arguments at different positions, and handle special characters in the input.
References: LPI Exam 101 Detailed Objectives, Topic 103: GNU and Unix Commands, Weight: 25, Objective 103.3: Perform basic file management, xargs command