DEVOPS INTERVIEW QUESTIONS &TASKS BY DEVSECOPS &DEVOPS SPECIALIST BOREDDY MANESH

TIME:10 hours

1. how to know the user is a root user or a sudo user

To know whether a particular user is having Sudo access or not, we can use **-l and -U options together**. For example, If the user has sudo access, it will print the level of sudo access for that particular user. If the user doesn't have sudo access, it will print that the user is not allowed to run sudo on localhost. How do I change to root user?

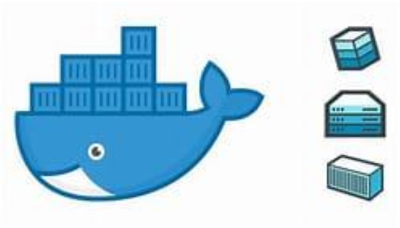
2. how to create users in Linux and how to give sudo permissions to the user

- Create a new Linux user. Step 1: Log in to your server as root. Step 2: Create a user using the useradd command. Replace the username with your custom user.
- Add Sudo Privileges to a User. Now let's make our new user or an existing user a sudo user. Step1: Add the user to the wheel group. ...
- Adding sudo privileges for specific command execution. There are scenarios where you might want only specific commands to be run Sudo privileges for a specific user.

3. how to display the last 10 commands which we have used in Linux

By using the history command

4. difference b/w docker file and docker-compose



The major difference between dockerfile and docker-compose is that docker-compose can run multiple containers from a single file, unlike the dotfiles file. Hence, make it easier to manage the taadministerinactors er actors. Another benefit of docker-compose is it provides higher security by isolating multiple containers over the same network.

5. explain about docker file

A Dockerfile is a text file that contains all the commands a user could run on the command line to create an image. It includes all the instructions needed by Docker to build the image.

6. explain docker volumes

Docker volumes are a widely used and useful tool for ensuring data persistence while working in containers. Docker volumes are file systems mounted on Docker containers to preserve data generated by the running container.

A Dockerfile is a text file that contains all the commands a user could run on the command line to create an image. It includes all the instructions needed by Docker to build the image.

7. how to restart a stopped container

Remind that when you restart a container it executes again its original command. So if you would able to restart the container of your use case (but you don't) it would run again /bin/bash -c "cat /tmp/cool-file" Restarting a container that runs with command /bin/bash, it will run again the same command when restarting.

Now, all we have left to do is pass the above command to the docker start,as shown below. One by one, all the container IDs will appear as Docker restarts them:

```
root@xps:~# docker start $(docker ps -a -q -f status=exited)

014a746dbb9d

080cf6412ac4
```

8. difference b/w VM and docker

Virtualization is the process of creating a virtual version of a server, desktop, operating system, storage device, or network resource. Docker uses container virtualization, whereas VM uses server virtualization. Docker is softwarere that provides a platform to execute applications. Moreover, it compacts different software components such as applications, tools, libraries, and configuration files into a complete standardized unit. On the other hand, VM is an operating system (OS) or application environment that is installed on software, which imitates dedicated hardware. Furthermore, it provides hardware-level virtualization.

9. why do we use ansible and explain about any ansible-playbook

Ansible is an open-source tool. Very simple to set up and use: No special coding skills are necessary to use Ansible's playbooks (more on playbooks later). Powerful Ansible lets you model even highly complex IT workflows. Flexible: You can orchestrate the entire application environment no matter where it's deployed.

Ansible contains a giant toolbox of built-in modules, well over 750 of them. Playbooks can finely **orchestrate multiple slices of your infrastructure topology**, with very detailed control over how many machines to tackle at a time. This is where Ansible starts to get most interesting.

10. what are Jenkins profiles and how to create

- **Step 1: Open your Jenkins dashboard by visiting http://localhost:8080/jenkins Steps 2: Click on 'Manage Jenkins' and select the 'Available' tab. Step 3:On the filter option, type "role-based" and press Enter. Step 4: Now, select the plugin and click on the 'Install without restart button. Step 5: Click on 'Go back to the top page'.**

11. how much do you know about maven and maven profiles

Maven Profiles allows to define multiple sets ofsetsnfigurations and activate them based on certain conditions. Maven Profile is an alternative set of configurations whichthator override default values. Profiles allow to cuscustomizingld for different environments.

Maven is a build automation tool used primarily for Java projects. Maven can also be used to build and manage projects written in C#, Ruby, Scala, and other languages. The Maven project is hosted by the Apache Software Foundation, where it was formerly part of the

12. what are the pipeline stages and explain about build stage

The steps that form a CI/CD pipeline are distinct subsets of tasks grouped into what is known as a pipeline stage. Typical pipeline stages include: **Build - The stage where the application is compiled**. Test - The stage where code is tested.

13. one user wants to process the only build step how do you give permissions the to a user to the do only build step for a particular job

14. explain about IAM and some databases in AWS

AWS Identity and Access Management (IAM) enables you to manage access to AWS services and resources securely. Using IAM, you can create and manage AWS users and groups, and use permissions to allow and deny their access to AWS resources. IAM is a feature of your AWS account offered at no additional charge. You will be charged only

Various databases that are provided by the Amazon Web Services (AWS) are :

- Amazon DynamoDB
- Amazon Aurora
- Amazon Relational Database Service
- Amazon Timestream
- Amazon Neptune
- Amazon Quantum Ledger Database (QLDB)
- Amazon RDS on VMware

15. explain about Kubernetes and what is the use and all

Kubernetes is a portable, extensible, open-source platform for **managing containerized workloads and services**, that facilitates both declarative configuration and automation. It has a large, rapidly growing ecosystem. Kubernetes services, support, and tools are widely available. The name Kubernetes originates from Greek, meaning helmsman or pilot.

16. namespace in docker

Namespace: Docker uses a technology called namespaces to provide the isolated workspace called the container. When you run a container, Docker creates a set of namespaces for that container. These namespaces provide a layer of isolation.

Docker Engine uses namespaces such as the following on Linux:

- The PID namespace: Process isolation (PID: Process ID).
- The net namespace: Managing network interfaces (NET: Networking).
- The IPC namespace: Managing access to IPC resources (IPC: InterProcess Communication).
- The mnt namespace: Managing filesystem mount points (MNT: Mount).
- The uts namespace: Isolating kernel and version identifiers. ...

17. 3 containers are running, one container is consuming all the CPU memory (99%) will it be caused

any with the in issue the to remaining containers? If yes what is the solution?

The --memory parameter limits the container memory usage, and Docker will **kill the container** if the container tries to use more than the limited memory. But inside the container, you still see the whole system's available memory. free reports the available memory, not the allowed memory.

18. We are running on one container and that got failed. What are the possible ways to rectify that

and what are the steps to take further

19. How to check whether the particular process is running properly or not? in Linux

1. Open the terminal window on Linux.
2. For remote Linux servers use the ssh command for login purposes.
3. Type the ps aux to see all running processes in Linux.
4. Alternatively, you can issue the top command or top command to view the running process in Linux.

The easiest way to find out if the process is running runinto**h ps aux command and grep process name**. If you got output along with procesPIDame/pid, PIDr process is running. Find out pPIDess pid TPID the following ps command to display all rprocprocesses process

20. I want to stop particular processes using Linux commands

1. What Processes Can You Kill in Linux?
2. View Running Linux Processes The top command is the easiest way to get a complete overview of the processes currently being run. ...
3. Locate the Process to Kill Before you can kill a process, you need to find it. There are multiple ways you can search for a process in Linux. ...
4. Use Kill Command Options to Terminate a Process

21. What are all branches u have used in Jenkins

22. If the build fails what are all the steps we should take

The Build Failure Analyzer Jenkins plugin analyzes the causes of failed builds and presents the causes on the build page. It does this by using a knowledge base of build failure causes maintained from scratch.

23. SED command completely

- SED is a powerful text stream editor. Can do insertion, deletion, search and replace(substitution).
- SED command in Unix supports regular expression which allows it to perform complex pattern matching.

Syntax:

```
sed OPTIONS... [SCRIPT] [INPUTFILE...]
```

24. Can we delete content in a file by using the SED command

There is no available to delete all contents of the file. How to delete all contents of the file using the sed command.

25. How the automatic tests happen in Jenkins

Jenkins - Automated Deployment. There are many plugins available that can be used to transfer the build files after a successful build to the respective application/webserver. example is the “Deploy to container Plugin”. To use this follow the steps given below. Step 1 − Go to Manage Jenkins → Manage Plugins.

26. Docker build option description

The `docker build` command builds Docker images from a Dockerfile and a “context”. A build’s context is the set of files located in the specified `PATH` or `URL`. The build process can refer to any of the files in the context. For example, your build can use a *COPY* instruction to reference a file in the context.

27. We have a file called abc.txt in our system, how to check, in which location that file exists using the Linux command

To obtain the full path of a file, we **use the readlink command**. reading prints the absolute path of a symbolic link, but as a side-effect, it also prints the absolute path for a relative path. In the case of the first command, reading resolves the relative path of foo/ to the absolute path of /home/example/foo/.

28. Tell the branch names you have worked in GIT

Determine the current branch name in Git

- git-branch. We can use the --show-current option of the git-branch command to print the current branch's name. ...
- git-rev-parse. Another plausible way of retrieving the name of the current branch is with git-rev-parse. ...
- git-symbolic-ref. ...
- git-name-rev.

29. Suppose we have a java.exe file and we have to run as a container, write a docker file and

mention all the variables which we need to execute java.exe

30. Day to day activities on CI/CD

I am penning down some of my thought from my own experience as well some studies I have done. Please review.

1. **Make sure that the pipeline is running smoothly –** This is one of the most important tasks of a DevOps engineer to make sure that the CI/CD pipeline is intact and fixing any issue or failure with it is the #1 priority for the day. They often need to spend time on troubleshootianalyzing sin,gingng and providing fixes to issues.
2. **Interaction with other teams** – Co-ordination and collaboration is the key for DevOps to be successful and hence daily integration with Dev and QA team, Program Management, and IT is always required.
3. **Work on Automation Backlog** – Automation is the soul of DevOps so DevOps engineering need to plan it out and I can

see DevOps engineer spending lots of time behind the keyboard working on Automating stuff on daily basis.

4. **Infrastructure Management** – DevOps engineers are also responsible for maintaining and managing the infrastructure required for the CI/CD pipeline and making sure it satit'ss up and running and being used optimally is also part of their daily schedule. Ex. Working on Backup, High Availability, NePlatformor,m se,tup etc.
5. **Dealing with Legacy stuff** – Not everyone is lucky to work on the latest and newest things and DevOps engineers are no exception hence they also need to spend time on legacy i.e. in terms of supporting it or migrating to the latest.
6. **Exploration** – DevOps leverage a lot from the available various tools, there are many options as open-source so the tneeds the to regularly check on this to make sure the adoptions asrequiretireded, this is something which also rerequiresome effot ,n ot on daily but regular basis. Ex. What are open source options available to keep the cost at a minimum?
7. **Removing bottlenecks** –DevOp'ss primary purpose is to identify the bottlenecks / Manual handshakes and work with everyone involved (Dev / QA and all othstakeholdersder) to remove them th so spends spend ga ood amount of tiin finding such thingsbuildingbuild the Automation Backlog using this. Ex. How we can get builds faster?
8. **Documentation** – Though Agile / DevOps stresses less on the documentation, it is still the important one that DevOps engineer does on daily basis, Be it Server Information, Daily Week charted, Scrum / Kanban board,o r Simple steps to configure / backup or modify the infrastructure, you need to spend a good amount of time in coming up these artifacts.
9. **Training and Self Development** – Self leaning and Training is very useful in getting a better understanding and manorganizationsns encourage their employee to take the time out and do some of these anthe d same true for DevOps folks as well, So learn something new every day…
10. **Continuous Improvement as Practice** – Last but not least, It's up to the DevOps folks to build awareness of the potential of CI/CD and DevOps practices and build a culture of leveraging it for doing things better, reducing re-work, increasing the productivity and optoptimizinge use of existing

resources. Go and talk to people to build the DevOps and Continuous Improvement culture...

31. Why do we use docker containerization

Containerization is **the packaging together of software code with all its necessary components like libraries, frameworks, and other dependencies so that they are isolated in their own "**

Containerization **allows developers to create and deploy applications faster and more securely**.

Hey Manesh

Some more interview questions.

1. What is docker why do we are using docker difference between VM?

   Docker is an open-source containerization platform. It **enables developers to package applications into containers—**standardized executable components combining application source code with the operating system (OS) libraries and dependencies required to run that code in any environment

   In Docker, the containers running share the host OS kernel. A Virtual Machine, on the other hand, is not based on container technology. They are made up of user space plus kernel space of an operating system. Under VMs, the server hardware is virtualized.
2. What is the docker cloud & how it is different from the docker hub what are the features of the docker hub?

   Docker Cloud uses Docker Hub as its native registry for storing both public and private repositories. Once you push your images to Docker Hub, they will be available in Docker Cloud. Additional reading: Overview of Docker Cloud.

   What is docker-compose?

   Docker Compose is **a tool that was developed to help define and share multi-container applications**. With Compose, we can create a YAML file to define the services and with a single command, can spin everything up or tear it all down.

4. How you will link when the docker containers are in a different virtual machine is there any

configuration in the docker-compose file is any command or any variable?

1. Open Oracle VM VirtualBox Manager
2. Select the VM used by Docker
3. Click Settings -> Network
4. Adapter 1 should (default?) be "Attached to NAT"
5. Click Advanced -> Port Forwarding
6. Add rule: Protocol TCP, Host Port 8080, Guest Post 8080 (leave Host IP and Guest IP empty)
7. Guest is your docker container and Host is your machine

Docker Compose file is **a tool that was developed to help define and share multi-container applications**. With Compose, we can create a YAML file to define the services and with a single command, can spin everything up or tear it all down.

5. What you will do if one master got corrupted,?

6. What you will do in case any pod is deleted?

if you manually deploy a single pod and then delete it, **your service will go down and won't come back up**. If a service is running through a replica set but with only one pod, the service will become unavailable after deleting the pod

8. What are namespaces in Kubernetes can you tell me some?

Namespaces are **a way to organize clusters into virtual sub-clusters** — they can be helpful when different teams or projects share a Kubernetes cluster. Any number of namespaces are supported within a cluster, each logically separated from others but with the ability to communicate with each other.

9. Can you tell me some commands using Kubernetes?

Kubectl controls the Kubernetes Cluster. It is one of the key components of Kubernetes which runs on the workstation on any machine when the setup is done. It can manage the nodes in the cluster.

Kubectl commands are used to interact and manage Kubernetes objects and the cluster. In this chapter, we will discuss a few commands used in Kubernetes via kubectl.

kubectl annotate − It updates the annotation on a resource.

```
$kubectl annotate [--overwrite] (-f FILENAME | TYPE NAME)
KEY_1=VAL_1 ...
KEY_N = VAL_N [--resource-version = version]
```

For example,

```
kubectl annotate pods tomcat description = 'my frontend'
```

kubectl api-versions − It prints the supported versions of API on the cluster.

```
$ kubectl api-version;
```

kubectl apply − It has the capability to configure a resource by file or stdin.

```
$ kubectl apply –f <filename>
```

kubectl attach − This attaches things to the running container.

```
$ kubectl attach <pod> –c <container>
$ kubectl attach 123456-7890 -c tomcat-conatiner
```

kubectl autoscale − This is used to auto scale pods which are defined such as Deployment, replica set, Replication Controller.

```
$ kubectl autoscale (-f FILENAME | TYPE NAME | TYPE/NAME) [--min =
MINPODS] --
max = MAXPODS [--cpu-percent = CPU] [flags]
$ kubectl autoscale deployment foo --min = 2 --max = 10
```

kubectl cluster-info − It displays the cluster Info.

```
$ kubectl cluster-info
```

kubectl cluster-info dump − It dumps relevant information regarding cluster for debugging and diagnosis.

```
$ kubectl cluster-info dump
$ kubectl cluster-info dump --output-directory = /path/to/cluster-
state
```

kubectl config − Modifies the kubeconfig file.

```
$ kubectl config <SUBCOMMAD>
$ kubectl config –-kubeconfig <String of File name>
```

kubectl config current-context − It displays the current context.

```
$ kubectl config current-context
#deploys the current context
```

kubectl config delete-cluster − Deletes the specified cluster from kubeconfig.

```
$ kubectl config delete-cluster <Cluster Name>
```

kubectl config delete-context − Deletes a specified context from kubeconfig.

```
$ kubectl config delete-context <Context Name>
```

kubectl config get-clusters − Displays cluster defined in the kubeconfig.

```
$ kubectl config get-cluster
$ kubectl config get-cluster <Cluser Name>
```

kubectl config get-contexts − Describes one or many contexts.

```
$ kubectl config get-context <Context Name>
```

kubectl config set-cluster − Sets the cluster entry in Kubernetes.

```
$ kubectl config set-cluster NAME [--server = server] [--
certificateauthority =
path/to/certificate/authority] [--insecure-skip-tls-verify = true]
```

kubectl config set-context − Sets a context entry in kubernetes entrypoint.

```
$ kubectl config set-context NAME [--cluster = cluster_nickname]
[--
user = user_nickname] [--namespace = namespace]
$ kubectl config set-context prod –user = vipin-mishra
```

kubectl config set-credentials − Sets a user entry in kubeconfig.

```
$ kubectl config set-credentials cluster-admin --username = vipin
--
password = uXFGweU9l35qcif
```

kubectl config set − Sets an individual value in kubeconfig file.

```
$ kubectl config set PROPERTY_NAME PROPERTY_VALUE
```

kubectl config unset − It unsets a specific component in kubectl.

```
$ kubectl config unset PROPERTY_NAME PROPERTY_VALUE
```

kubectl config use-context − Sets the current context in kubectl file.

```
$ kubectl config use-context <Context Name>
```

kubectl config view

```
$ kubectl config view
$ kubectl config view –o jsonpath='{.users[?(@.name ==
"e2e")].user.password}'
```

kubectl cp − Copy files and directories to and from containers.

```
$ kubectl cp <Files from source> <Files to Destinatiion>
$ kubectl cp /tmp/foo <some-pod>:/tmp/bar -c <specific-container>
```

kubectl create − To create resource by filename of or stdin. To do this, JSON or YAML formats are accepted.

```
$ kubectl create –f <File Name>
$ cat <file name> | kubectl create –f -
```

In the same way, we can create multiple things as listed using the **create** command along with **kubectl**.

- deployment
- namespace
- quota
- secret docker-registry
- secret
- secret generic
- secret this
- service accountant
- service cluster
- service load balancer
- service deport

kubectl delete − Deletes resources by file name, stdin, resource, and names.

```
$ kubectl delete –f ([-f FILENAME] | TYPE [(NAME | -l label | --
all)])
```

kubectl describe − Describes any particular resource in Kubernetes. Shows details of resources or a group of resources.

```
$ kubectl describe <type> <type name>
$ kubectl describe pod tomcat
```

kubectl drain − This is used to drain a node for maintenance purposes. It prepares the node for maintenance. This will mark the node as unavailable so that it should not be assigned with a new container that will be created.

```
$ kubectl drain tomcat –force
```

kubectl edit − It is used to end the resources on the server. This allows one to directly edit a resouthathich one can receive via the commalineine tool.

```
$ kubectl edit <Resource/Name | File Name)
Ex.
$ kubectl edit rc/tomcat
```

kubectl exec − This helps to execute a command in the container.

```
$ kubectl exec POD <-c CONTAINER > -- COMMAND < args...>
$ kubectl exec tomcat 123-5-456 date
```

kubectl expose − This is used to expose the Kubernetes objects such as pod, replication controller, and service as a new Kubernetes service. This can expose it via a running container or from a **yaml** file.

```
$ kubecYAMLxpose (-f FILENAME | TYPE NAME) [--port=port] [--
protocol = TCP|UDP]
[--target-port = number-or-name] [--name = name] [--external-ip =
external-ip-ofservice]
[--type = type]
$ kubectl expose rc tomcat –-port=80 –target-port = 30000
$ kubectl expose –f tomcat.yaml –port = 80 –target-port =
```

kubectl get − This command is capable of fetching data on the cluster about the Kubernetes resources.

```
$ kubectl get [(-o|--output=)json|yaml|wide|custom-
columns=...|custom-columnsfile=...|
go-template=...|go-template-file=...|jsonpath=...|jsonpath-
file=...]
(TYPE [NAME | -l label] | TYPE/NAME ...) [flags]
```

For example,

```
$ kubectl get pod <pod name>
$ kubectl get service <Service name>
```

kubectl logs − They are used to get the logs of the container in a pod. Printing the logs can be defining the container name in the pod. If the POD has only one container there is no need to define its name.

```
$ kubectl logs [-f] [-p] POD [-c CONTAINER]
Example
$ kubectl logs tomcat.
$ kubectl logs –p –c tomcat.8
```

kubectl port-forward − They are used to forward one or more local port to pods.

```
$ kubectl port-forward POD [LOCAL_PORT:]REMOTE_PORT
[...[LOCAL_PORT_N:]REMOTE_PORT_N]
$ kubectl port-forward tomcat 3000 4000
$ kubectl port-forward tomcat 3000:5000
```

kubectl replace − Capable of replacing a resource by file name or **stdin**.

```
$ kubectl replace -f FILENAME
$ kubectl replace –f tomcat.yml
$ cat tomcat.yml | kubectl replace –f -
```

kubectl rolling-update − Performs a rolling update on a replication controller. Replaces the specified replication controller with a new replication controller by updating a POD at a time.

```
$ kubectl rolling-update OLD_CONTROLLER_NAME
([NEW_CONTROLLER_NAME] --
image = NEW_CONTAINER_IMAGE | -f NEW_CONTROLLER_SPEC)
$ kubectl rolling-update frontend-v1 –f freontend-v2.yaml
```

kubectl rollout − It is capable of managing the rollout of deployment.

```
$ Kubectl rollout <Sub Command>
$ kubectl rollout undo deployment/tomcat
```

Apart from the above, we can perform multiple tasks using the rollout such as −

- rollout history
- rollout pause
- rollout resume
- rollout status
- rollout undo

kubectl run − Run command has the capability to run an image on the Kubernetes cluster.

```
$ kubectl run NAME --image = image [--env = "key = value"] [--port
= port] [--
replicas = replicas] [--dry-run = bool] [--overrides = inline-
json] [--command] --
[COMMAND] [args...]
$ kubectl run tomcat --image = tomcat:7.0
$ kubectl run tomcat --image = tomcat:7.0 –port = 5000
```

kubectl scale − It will scale the size of Kubernetes Deployments, ReplicaSet, Replication Controller, or job.

```
$ kubectl scale [--resource-version = version] [--current-replicas
= count] --
```

```
replicas = COUNT (-f FILENAME | TYPE NAME )
$ kubectl scale --replica = 3 rs/tomcat
$ kubectl scale –replica = 3 tomcat.yaml
```

kubectl set image − It updates the image of a pod template.

```
$ kubectl set image (-f FILENAME | TYPE NAME)
CONTAINER_NAME_1 = CONTAINER_IMAGE_1 ... CONTAINER_NAME_N =
CONTAINER_IMAGE_N
$ kubectl set image deployment/tomcat busybox = busybox ngnix =
ngnix:1.9.1
$ kubectl set image deployments, rc tomcat = tomcat6.0 --all
```

kubectl set resources − It is used to set the content of the resource. It updates resource/limits on object with pod template.

```
$ kubectl set resources (-f FILENAME | TYPE NAME) ([--limits =
LIMITS & --
requests = REQUESTS]
$ kubectl set resources deployment tomcat -c = tomcat --
limits = cpu = 200m,memory = 512Mi
```

kubectl top node − It displays CPU/Memory/Storage usage. The top command allows you to see the resource consumption for nodes.

```
$ kubectl top node [node Name]
```

10. Tell me the command to create a cluster?

The **cluster creation** command creates a cluster with one node. Once you create the cluster, add additional nodes to the cluster by using the cluster join command. Note that single-node clusters do not require configuring the cluster network.

Difference between RC and rs?

Replica Set and Replication Controller do almost the same thing. **Both of them ensure that a specified number of pod replicas are running at any given time**. The difference comes with the usage of selectors to replicate pods. Replica Set uses Set-Based selectors while replication controllers use Equity-Based selectors.

11. What is Kubernetes?

Kubernetes **smoothens the container tasks**. It eases activities like canary deployment, rolling updates, and horizontal auto-scaling. It plays a major role in streamlining the development, testing as well as deploying pipelines in the DevOps Managed Services.

12 What is the difference between kubctl & kops?

Kubernetes Operations, or Kops, is an open-source project used to set up Kubernetes clusters easily and swiftly. It's considered the “kubectl” way of creating clusters. Kops allows deployment of highly available Kubernetes clusters on AWS and Google

Kubectl is **a command-line tool used to run commands against Kubernetes clusters**. It does this by authenticating with the Master Node of your cluster and making API calls to do a variety of management actions

13 Why you are using kubectl can you explain why we are using it?

14 Why are we using kops?

15 What is the difference between docker cloud and docker swarm?

Docker Cloud is **a native Docker solution designed to deploy and manage Dockerized applications**. With native integrations to Docker tools like Docker Hub and Docker Engine, Docker

Cloud simplifies the entire process of provisioning nodes, installing Docker Engine, and connecting to image repos on Docker Hub.

Docker Swarm is a lightweight, easy-to-use orchestration tool with limited offerings compared to Kubernetes

16 **How to attach a volume in a cluster at some time the container will be deleted then rs will recreate a new container then how to attach that container automatically and how to restore the volume automatically to the re-created container**?

17 How many projects have you used Kubernetes?

18 How many projects have you used Kubernetes?

19 If installed Kubernetes how you will deploy this container into the Kubernetes cluster?

20 C**an you tell me the command for creating a Kubernetes cluster in VM?**

- Using vagrant spin up multiple virtual machines
- Install Kubernetes and set up Controller(master) and Workers
- Launch an application scale up/down and expose it.
- containersrd as the container runtime

21 **How many nodes are we required to create a Kubernetes cluster**?

5000 nodes

A cluster is a set of nodes (physical or virtual machines) running Kubernetes agents, managed by the control plane. Kubernetes v1. 23 supports clusters with **up to 5000 nodes**.

22 We have nearly 15 nodes in my organization all are decentralized so which node do I need to create

as a master? Is there any possibility to make all the machines masters?

23 Our applications are decentralized I don't want distributed environment anything thing happens too

the master all will collapse, can we create multiple masters?

24 What is the difference between kubectl and minikube?

Kubernetes is an open-source orchestration system for Docker containers. It handles scheduling onto nodes in a compute cluster and actively manages workloads to ensure that their state matches the users declared intentions. On the other hand, **minikube is detailed as a "Local Kubernetes engine"**.

25 **If any container down in my cluster how you will rectify it?**

26 How can I print the shell script name?

Echo $0

27 How to write a script when the first command is executed then execute the below script?

```
A) Java --version

If [ $? –eq 0 ]

Then

Echo “print the variable ”

Else

Echo “prin the variable”

fi
```

28 How to print the exact file name by using the command?

A) First assign path as a variable

```
awk -F '/' '{print $(NF-1)}' <<< "$a"
```

29 Can you tell me the syntax for loop and while loop?

30 How to dictionary in python?

A) Dict={a:10,b:10}

31 How to add another variable c, with key 10 to the above dictionary?

32 How to replace an existing dictionary?

A) Dict=[c:10]

33 **How to print shell name**?

1. 2.1. echo $SHELL. The $SHELL variable contains the name of the default shell. We can display its value: ...
2. 2.2. echo $0. We can also use the echo command with the $0 variable: $ echo $0 bash. This approach works well on the command line but not from within a script.

34 How to assign all the arguments to a single variable?

35 **How to print the current processid of the current shell?**

1. Open the terminal application.
2. Run your command or app in the background. For example firefox &
3. To get the PID of the last executed command type: echo "$!"
4. Store the PID of the last command in a variable named foo: foo=$!
5. Print it, run: echo "$foo"

36 How to know the file that is entering randomly into my script?

3**7How to divide two variables in a shell script?**

Shell script for division of two numbers

1. initialize two variables.
2. divide two numbers directly using $(...) or by using external program expr.
3. Echo the final result.

38 What is a trap?

trap **defines and activates handlers to run when the shell receives signals or other special conditions**. ARG is a command to be read and executed when the shell receives the signal(s) SIGNAL_SPEC

39 **What is a shift in a shell script?**

The shift built-iinline**command in bash which after getting executed, shifts/move the command line arguments to one position left**. The first argument is lost after uthesing shift command. This command takes only one integer as an argument.

40 **How to run our script in the background?**

Running shell command or script in the background using **nohup command**. Another way you can run a command in the background is using the nohup command. The nohup command, short forhang-upg up, is a command that keeps a process running even after exiting the shell

41 **How to know the running background process id?**

The easiest way to find out if the process is runningto is **run ps aux command and grep process name**. If you got output along with procesPIDame/pid, your process is running.

42 What are $*, $,$ and $@?

$0 Stores the first word of the entered command (the name of the shell program). $* Stores all the arguments that were entered on the command line ($1 $2 ...). "$@" Stores all the arguments that were entered on the command line, individually quoted ("$1" "$2" ...).

43 **How to print only directories**?

You can **use a combination othe f ls command, find command, and grep command to list directory names only**.

44 **How to print the directory only started with a number?**

45 How to grep two strings at a time?

46 How to grep a string that is started with some string and ends with some string like a…..b?

47 How to print a string that starts with a?

To print a string in Bash, **use the echo command**. Provide the stringa as command line argument to echo command.

48 Did you work on arrays?

1. Give your array a name.
2. Follow that variable name with an equal sign. The equal sign should not have any spaces around it.
3. Enclose the array in parentheses (not brackets like in JavaScript)
4. Type your strings using quotes, but with no commas between them.

49 How will you give access to your script to a particular user?

To change file and directory permissions, **use the command chmod (change mode)**. The owner of a file can change the permissions for user (u), group (g), or others (o) by adding (+) or subtracting (-) the read, write, and execute permissions.

50 How to access background running scripts and they're paid?

How do I see running processes in Linux?

1. Open the terminal window on Linux.

2. For remote Linux servers use the ssh command forloginn purposes.
3. Type the ps aux command to see all running processes in Linux.
4. Alternatively, you can issue the top command or top command to view rthe unning process in Linux.

51 How to run our script in the foreground?

1. To run the count program, which will display the process identification number of the job, enter: count &
2. To check the status of your job, enter jobs.
3. To bring a background process to the foreground, enter: FG.
4. If you have more than one job suspended in the background, enter: FG %#

Ansible interview questions:

1. What is the inventory file used for and the default inventory host location?

An inventory file is **a document containing listings, usually electronic, of every item in a company's inventory, including items in stock or expected to be in stock shortly**.

/etc/ansible/hosts

The default location for the inventory file is **/etc/ansible/hosts**. You can also create project-specific inventory files in alternate locations.

2. What is an ansible configuration file used for and its default path?

With a fresh installation of Ansible, like every other software, it ships with a default configuration file. This is the brain and the heart of Ansible, **the file that governs the behavior of all interactions performed by the control node**

The default Ansible configuration file is located under **/etc/ansible/ansible. cfg**

3. Do you write your inventory file?

Ansible uses an inventory file to keep track of which hosts are part of your infrastructure, and how to reach them for running commands and playbooks. There are multiple ways in which you can set up your Ansible inventory file, depending on your environment and project needs.

Let's see how to set an inventory file when you run a playbook. Ansible works with an inventory file. It contains a list of hosts. You then tell Ansible to run a playbook on the hosts in the inventory file. An inventory file might look like this: By default, Ansible will look in /etc/ansible/hosts for an inventory file.

4. How many types of variables and precedence?

Ansible supports **two types of inventory variables** Host Variables and Group Variables. Host variables are applied to a specific host for which the variable is declared. On the other hand, Group Variables are applied to a group of hosts

Ansible offers four sources for controlling its behavior. In order of precedence from lowest (most easily overridden) to highest (overrides all others), the categories are:

- Configuration settings

- Command-line options
- Playbook keywords
- Variables

5. Write the command to find the python version on nodes?

6. What is the file structure of Ansible roles?

An Ansible role is composed of **multiple folders, each of which contains several YAML files**. By default, they have a **main.yml** file, but they can have more than one when needed. This is a standardized structure for all Ansible roles, which allows Ansible playbooks to automatically load predefined variables, tasks, handlers, templates, and default values located in separate YAML files.

What happens when one node or instance is unreachable?

If Ansible cannot connect to a host, it marks that host as 'UNREACHABLE' and **removes it from the list of active hosts for the run**. You can use meta: clear_host_errors to reactivate all hosts, so subsequent tasks can try to reach them again.

8. What happens when one task is failed in the playbook?

When that happens, the playbook fails and you have to run it again to get back to the point where you can start configuring things again.

9. I have 20 servers, I want to install one package on 5 servers and the other package in the next 5 servers..like that How to write in ansible script for that…Explain?

10. What is the architecture of Ansible?

Ansible Architecture

- Inventory. Inventory is list of nodes or hosts having their IP addresses, databases, servers, etc. ...
- apis's. The Ansible API woworks thensport for the pubor private cloud services.
- Modules. Ansible connected the nodes and spread out the Ansible modules programs. ...
- Plugins. ...
- Playbooks. ...

How do I supply variables while executing the playbook in ansible?

You can define variables when you run your playbook by passing variables at the command line **using the --extra-vars (or -e) argument**

12. Explain about the tags in ansible?

A tag is an attribute that you can set to an Ansible structure (plays, roles, tasks), and then when you run a playbook you can use –tags or –skip-tags to execute a subset of tasks

13. How to execute the failure playbook again?

1. ansible-playbook ./main.YAML
2. Playbook fails on some task

3. Fix this task and repeat line 1, waiting for all previous tasks to execute again. Which takes a lot of time

14. How is ansible different from any other configuration management tool?

it has 'agentless' architecture. Chef and Puppet follow master-agent or master-slave architecture

15. Ansible playbooks are written in what format?

YAML

Ansible playbooks are written in **YAML**, YAML Ain't Markup Language. Understanding YAML syntax is a key to success with Ansible. If you write or use Ansible playbooks, then you're used to reading YAML configuration files

16. What is a module in Ansible?

Ansible modules are **standalone scripts that can be used inside an Ansible playbook**. A playbook consists of a play, and a play consists of tasks. These concepts may seem confusing if you're new to Ansible, but as you begin writing and working more with playbooks, they will become familiar.

17. What is inventory used for in ansible?

The Ansible inventory file **defines the hosts and groups of hosts upon which commands, modules, and tasks in a playbook operate**. The file can be in one of many formats depending on your Ansible environment and plugins.

18. Name three places where ansible variables can be stored?

- variables in Playbooks.
- Variables with Arrays.
- Variables with dictionaries.
- Variables in Inventory files.
- Host and Group variables.
- Special variables in Ansible Playbook.

19. What connection is sensibly established with Linux and Windows nodes.

1. Configure SSHD. The Linux system (Fedora 33 in my case) acts as the SSH server that allows the PuTTY SSH client to connect. ...
2. Set up a remote console. On Windows, download the PuTTY installer, then install and open it. ...
3. Copy files over the network. ...
4. Tunnel a protocol.

20. Difference between Remote and local execution in ansible.

21. What is the purpose and location of ansible.cfg file.

This is the brain and the heart of Ansible, **the file that governs the behavior of all interactions performed by the control node**

22. Name any two settings from ansible.cfg file.

- action_plugins. ...
- allow_unsafe_lookups. ...
- allow_world_readable_tmpfiles. ...
- ansible_managed. ...
- ask_pass. ...
- ask_sudo_pass. ...
- ask_vault_pass. ...
- bin_ansible_callbacks.

23. Please write below a sample inventory file with host, group & group of groups syntax in it.

1. Step 1 — Creating a Custom Inventory File. ...
2. Step 2 — Organizing Servers Into Groups and Subgroups. ...
3. Step 3 — Setting Up Host Aliases. ...
4. Step 4 — Setting Up Host Variables. ...
5. Step 5 — Using Patterns to Target Execution of Commands and Playbooks.

24. What is the difference between group_vars & host_vars directory?

The host_vars is a similar folder to group_vars in the repository structure. It contains data models that apply to individual hosts/devices in the hosts. ini file. Hence, there is a YAML file created per device containing specific information about that device.

25. What are ad-hoc commands used for & write below the syntax of an ad hocs command?

Ad hoc commands are **commands which can be run individually to perform quick functions**. These commands need not be performed later. For example, you have to reboot all your company servers. For this, you will run the Adhoc commands from '/usr/bin/ansible

26. Write below ad hoc command to gather fact variables on all the hosts from the inventory file?

Ad hoc commands are **commands which can be run individually to perform quick functions**. These commands need not be performed later. For example, you have to reboot all your company servers. For this, you will run the Adhoc commands from '/usr/bin/ansible'.

27. What format does the ansible ad hoc command returns the output?

Use the -o option to display the output of Ansible ad hoc commands in a **single-line format**.

28. Name three types of modules in ansible?

Ansible module architecture

- Action plugins.
- New-style modules. Python. PowerShell.
- JSONARGS modules.
- Non-native want JSON modules.
- Binary modules.
- Old-style modules.
- * ...
- Service Module. ...
- Debug Module. ...
- Template Module.

29. Name any 10 modules in ansible that you have used.

- Ping Module. Ping is used when we want to check whether the connection with our hosts defined in the inventory file is established or not. ...
- Setup Module. ...
- Copy Module. ...
- Yum Module. ...

Shell Module

30. How to list all the Ansible core modules from the command line.

Ansible has a very attractive command named **ansible-doc**. This command will tell all the module details installed in your system.

31. How to display all the options/attributes for the apt module from the command line.

This **module** is part of ansible-core and included in **all** Ansible installations. In most cases, you can use the short **module** name **apt** even .

32. How can you check the mandatory option for any module from the command line?

You can access the documentation for each module **from the command line with the ansible-doc tool**. For a list of all available modules, see the Collection docs, or run the following at a command prompt

33. What is the setup module used for?

to fetch the facts about the system, and further, it will use the filter argument to display the value from the Ansible facts. Note: Ansible facts are retrieved only when working with playbooks. To access the Ansible facts using ad-hoc commands, use the setup module.

34. Write down a sample global play declaration

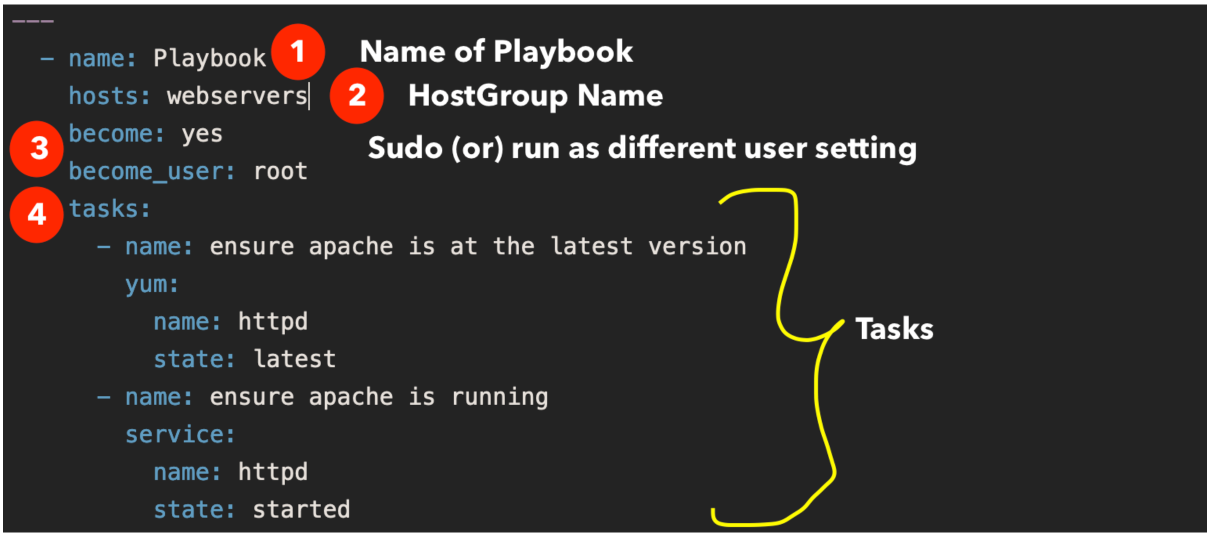
1. Global: this is set by config, environment variables, and the command line.
2. Play: each play contained structures, vars entries, include_vars, role defaults and, vars.
3. Host: variables directly associated with a host, like inventory, fac,ts or registered task outputs.

35. Write down any two tasks from the playbook with their proper format and names.

Ansible Playbook Example

```
---

  - name: Playbook

    hosts: webservers

    become: yes

    become_user: root
    tasks:

      - name: ensure apache is at the latest version

        yum:

          name: httpd

          state: latest

      - name: ensure apache is running

        service:

          name: httpd

          state: started
```

this simple ansible-playbook example given above is enough to get your Apache installation done and ready. I sense your anger that I just gave a plain text with no explanation of what they do.



```
---
  - name: Playbook
    hosts: webservers
    become: yes
    become_user: root
    tasks:
      - name: ensure apache is at the latest version
        yum:
          name: httpd
          state: latest
      - name: ensure apache is running
        service:
          name: httpd
          state: started
```

Well, I have explained what each line does

`name` Name of the playbook

`hosts` A set of hosts is usually grouper as a host group and defined in inventthe ory file

`become` To tell ansible this play has to be executed with elevated privileges `become_user` the user name that we want to switch to like compare it with `Sudo su - user`

`tasks` set of tasks to execute, All tasks would be defined below this

and then we have two tasks with two modules, the first module is `yum` and the second module is `service`

in the first task with yum the state `latest` represents that the forementioned package `httpd` should be installed if it is not installed (or) if it is already installed it should be upgraded to the latest version available. If you do not want it to be upgraded if present, You can change it to `state: present`

On the Second task with the service module, we are making sure that the service named `httpd` is started and running using the `state: started` Ansible would not restart the service if it is already started and running.

36. What is the difference between Sudo and become module and its purpose

become_user defines the user which is being used for privilege escalation.

become simply is a flag to either activate or deactivate the same.

37. Write the down playbook syntax of starting NTP service on webserver and observers host group at

once.

38. How to take user input from a playbook?

If you want your playbook to prompt the user for a certain input, **add a 'vars_prompt' section**. Prompting the user for variables lets you avoid recording sensitive data like passwords. In addition to security, prompts support flexibility.

39. What is debug module used for in playbooks?

Ansible provides a debug module option that **makes the tasks more manageable**. It is a handy tool to figure out any problem areas. Ansible version 2.1 extended the debug module with a verbosity parameter that transforms it from a print line.

40. How to store the output of any task into a variable from the playbook?

Create the playbook to execute the “df” command to check the /boot usage. **Use “register” to store the output to a variable**. 2. Run the playbook to see the result.

41. What are handlers used for in ansible and how is it different from tasks?

Handlers are just like regular tasks in an Ansible playbook (see Tasks) but are only run if the Task contains a notify keyword and also indicates that it changed something. For example, if a config file is changed, then the task referencing the config file templating operation may notify a service restart handler.

42. Conditional execution in ansible is used for what purpose and write down its syntax with small description?

43. What are templates used for and their format?

Templates are **pre-formatted documents, intended to speed up the creation of commonly used document types such as letters, fax forms, or envelopes**. Templates are also used as guidelines for creating documents in a specific format

44. What are ansible roles and their purpose?

Ansible roles **allow you to develop reusable automation components by grouping and encapsulating related automation artifacts, like configuration files, templates, tasks, and handlers**. Because roles isolate these components, it's easier to reuse them and share them with other people

45. Command to generate ansible roles directory structure.

To create an ansible role, **use ansible-galaxy init <role_name>** to create the role directory structure

46. Name 5 directories from ansible roles.

A role directory structure contains directories: **defaults, vars, tasks, files, templates, meta, handlers**

60. Write an Ansible task that can copy the file to a remote location with the ownership of Jboss?

1. name: Ansible Copy Example Local to Remote.
2. hosts: remote server.
3. tasks:
4. - name: copying file with playbook.
5. become: true.
6. copy:
7. src: ~/Downloads/index.html.
8. dest: /var/www/HTML.

61. What is local action in Ansible?

In an Ansible playbook, when local_action is used, **Ansible will run the module work mentioned under it on the controller node**. Most modules used with local_action are shell and command. As you can do almost all the tasks using these modules on the controller node.

62. What are the roles in Ansible?

Roles **let you automatically load related vars, files, tasks, handlers, and other Ansible artifacts based on a known file structure**. After you group your content in roles, you can easily reuse them and share them with other users. Role directory structure

63. Write a playbook for installation of apache in ubuntu and centos?

64. How to create a role in ansible?

1. Initialise the role structure using the command: ansible-galaxy init &lt;role-name&gt;
2. Go to role directory using: cd &lt;role-name&gt;

3. Initialise for Git: git init.
4. Do the necessary changes required to add role functionality.
5. Add files to Git using the command: git add *

65. Different types of inventories in ansible?

In Ansible, there are two types of inventory files: **Static and Dynamic**.

The dynamic inventory script can do anything to get the data (call an external API, pull information from a database or file, etc.), and Ansible will use it as an inventory source as long as it returns a JSON structure like the one above when the script is called with the –list

Under a static inventory management system (also called periodic inventory management), **inventory information must be updated by a regular physical count of each item in stock**. Stock takes can be annual but mmaybemuch more frequent whwhen business deals with a large quantity of inventory

66. if we want system information of machine how we will get the data with ansible as we use

factor in puppet?

67. In which language do we write playbooks in ansible?

68. Activity that we use with ansibles or advantage Functionality or purpose of ansible using in your project?

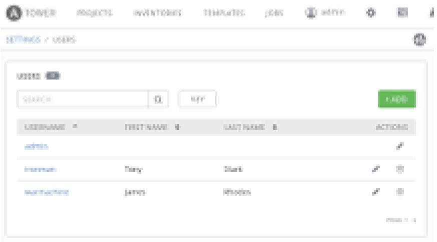
- Simple. Human readable automation. No special coding skills are needed. Tasks executed in order. Get productive quickly.
- Powerful. App deployment. Configuration management. Workflow orchestration. Orchestrate the app lifecycle.
- Agentless. Agentless architecture. Uses OpenSSH and WinRM. No agents to exploit or update.

Ansible is an open-source IT automation engine that **automates provisioning, configuration management, application deployment, orchestration, and many other IT processes**

69. How to launch an LDAP server using ansible?

The default port for LDAP is **port 389**, but LDAPS uses port 636 and establishes TLS/SSL upon connecting with a client.

How does Ansible tower integrate with LDAP?



USERNAME	FIRST NAME	LAST NAME	ACTIONS
admin			
ironman	Tony	Stark	
warmachine	James	Rhodes	

To configure your Ansible Tower for LDAP authentication, **navigate to Settings (the gear icon) and the "Configure Tower" section**.
...
CONFIGURATION SETTINGS

1. LDAP server URI.

2. Bind DN and password.
3. User/group searches.

70. How does ansible work.. and what is the playbook written other than the basic playbooks?

Ansible works **by connecting to your nodes and pushing out small programs, called modules to them**. Modules are used to accomplish automation tasks in Ansible. These programs are written to be resource models of the desired state of the system. Ansible then executes these modules and removes them when finished

71. For example there are 4 apps server running and 4 websites running and we have a new code

change and this change should automatically copy to these different servers using ansible, how I will come to know new code is generated.

Use Jenkins with POLL SCM option and after the artifact is generated use ansible to push suthe ing copy module.

How does Jenkins integrate with Ansible?

Jenkins Integration with Ansible

1. Step 1: Create a Jenkins job and configure the SCM repo using the code in GitHub.
2. Step 2: Configure the build.
3. Step 3: Create a roles directory within the Jenkins workspace.
4. Step 4: Create the tomcat role in the Jenkins workspace location using the command shown below.

72. Have u done any automation like the app goes down then for self-healing of the app kind of the

thing.

In software systems, the self-healing term describes any application, service, or a system that can discover that it is not working correctly and, without any human intervention, **make the necessary changes to restore itself to the normal or designed state**

73. Have u written any playbooks? Explain it

74. what is a dynamic inventory in ansible?

A dynamic inventory is **a script written in Python, PHP, or any other programming language**. It comes in handy in cloud environments such as AWS where IP addresses change once a virtual server is stopped and started again

75. How can you manage cloud services with ansible?

Ansible is an open-source tool that you can use to automate your AWS deployments. You can use it to define, deploy, and manage applications and services **using automation playbooks**. These playbooks enable you to define configurations once and deploy those configurations consistently across environments

76. How can you protect sensitive information in ansible?

Ansible Vault is a feature that allows users to encrypt values and data structures within Ansible projects. This provides the ability to secure any sensitive data that is necessary to successfully run Ansible plays but should not be publicly visible, like passwords or private keys

77. How can you save the output of module execution in ansible?

Logging Ansible output

1. To save Ansible output in a single log on the control node, set the log_path configuration file setting. ...
2. To save Ansible output in separate logs, one on each managed node, set the no_target_syslog and syslog_facility configuration file settings.

78. How can you manage error handling?

If you set any_errors_fatal and a task returns an error, Ansible finishes the fatal task on all hosts in the current batch, then stops executing the play on all hosts. Subsequent tasks and plays are not executed. You can recover from fatal errors by **adding a rescue section to the block**

79. what are conditionals in ansible?

When you add a conditional to an import statement, Ansible applies the condition to all tasks within the imported file. This behavior is the equivalent of Tag inheritance: adding tags to multiple tasks. Ansible applies the condition to every task and evaluates each task separately.

80. write a playbook to create AMI on AWS?

81. How can you the fact variables in ansible?

To access the variables from Ansible facts in the Ansible playbook, we need to **use the actual name without using the ansible keyword**. The gather_facts module from the Ansible playbook runs the setup module by default at the start of each playbook to gather the facts about remote hosts.

82. can we create an Ansible module?

If you need functionality that is not available in any of the thousands of Ansible modules found in collections, **you can easily write your custom module**. When you write a module for local use, you can choose any programming language and follow your own rules

Navigate to the correct directory for your new module: $ cd lib/ansible/modules/. If you are developing a module in a collection, $ cd plugins/modules/ inside your collection development tree. Create your new module file: $ touch my_test.py Paste the content below into your new module file

83 . How can you do provisioning with ansible?

Ansible can provision the latest cloud platforms, virtualized hosts and hypervisors, network devices, and bare-metal servers. **After bootstrapping, nodes can be connected to storage, added to a load balancer, security patche,d or any number of other operational tasks by separate teams**.

84 . BASH Vs Python Vs Configuration Management Tools?

85. **How can you update a single table in a Database with ansible?**

S.NO.	PYTHON	BASH
1	Python is a highly efficient programming language used for general-purpose programming.	Bash is not a programming language, it is a command-line interpreter.

S.NO.	PYTHON	BASH
2	Python is based on object-oriented programming	Bash is a software replacement for the original Bourne shell.
3	Python is an easy, sim,ple and powerful language.	Bash is tough to write and not powerful as python.
4	It is specially designed for web and app development.	It is found on Linux distributions and macOS.
5	Python is more efficient and is known for its consistency and readability.	IT does not deal with frameworks.
6	It supports OOP and allows users to easily and neatly break problems.	Bash does not support OOP and it only understands the t ext.L
7	It is easier to maintain than bash	It is harder to maintain as compared to python
8	It requiresthird-partyy programs to be installed	It does not require any third-party apps/programs to be installed
9	It is better to use python when the scriptlargerager thanLOC0 lOC.	For smaller scripts Bash is good.

Docker:

1. Current roles & responsibilities?

Some of the core responsibilities of a DevOps Engineer include –

- Understanding customer requirements and project KPIs
- Implementing various development, testing, automation tools, and IT infrastructure
- Planning the team structure, activities, and involvement in project management activities.
- Managing stakeholders and external interfaces
- Setting up tools and required infrastructure

- Defining and setting development, test, release, update, and support processes for DevOps operation
- Have the technical skill to review, verify, and validate the software code developed in the project.
- Troubleshooting techniques and fixing the code bugs
- Monitoring the processes during the entire lifecycle for their adherence and updating or creating new processes for improvement and minimizing the wastage
- Encouraging and building automated processes wherever possible
- Identifying and deploying cybersecurity measures by continuously performing vulnerability assessment and risk management
- Incidence management and root cause analysis
- Coordination and communication within the team and with customers
- Selecting and deploying appropriate CI/CD tools
- Strive for continuous improvement and build continuous integration, continuous development, and constant deployment pipeline (CI/CD Pipeline)
- Mentoring and guiding the team members
- Monitoring and measuring customer experience and KPIs
- Managing periodic reporting on the progress to the management and the customer

2. What is Docker composed of?

Compose is a tool for defining and running multi-container Docker applications. With Compose, you use **a YAML file** to configure your application's services. Then, with a single command, you create and start all the services from your configuration.

3. What is the Docker server version?

You can check the version of Docker you have installed with the following command from a terminal prompt:

1. docker --version.
2. sudo systemctl start docker.
3. sudo systemctl enable docker.
4. sudo usermod -a -G docker <username>
5. docker-compose --version.

4. What are the advantages of Docker?

Docker: Top 7 Benefits of Containerization

- Key Benefits of Docker Containers. ...

- Consistent and Isolated Environment. ...
- Cost-effectiveness with Fast Deployment. ...
- Mobility – Ability to Run Anywhere. ...
- Repeatability and Automation. ...
- Test, Roll Back and Deploy. ...
- Flexibility. ...
- Collaboration, Modularity, and Scaling

5. How do you set up Docker in Jenkins in a production environment?

Install the above two plugins using Jenkins' "Plugin Manager". Creating and configuring Jenkins job to build images from Dockerfile: **Create a new Jenkins job (say "Build Docker Image") which will use CloudBees Docker Build and Publish plugin to build images from Dockerfile and push it on DockerHub**.

6. **How do create a Docker image from the Docker file**?

Build the app's container image

1. Create a file named Dockerfile in the same folder as the file package. JSON with the following contents. ...
2. If you haven't already done so, open a terminal and go to the app directory with the Dockerfile. Now build the container image using the docker build command.

7. How do you deploy the Docker image generated in the testing environment to

The production environment in Jenkins?

In Jenkins, we have to create 3 jobs to automate the integration and deployment process.

We've created a git repository and created a sample HTML file for testing the setup, and uploaded it in the git hub using the Git Push command. And created a copy of the file in a Git Branch namely Dev1 for the development and testing.

We need a webserver to launch the sample application we made. here we use the Apache HTTPD server using the Docker httpd image.

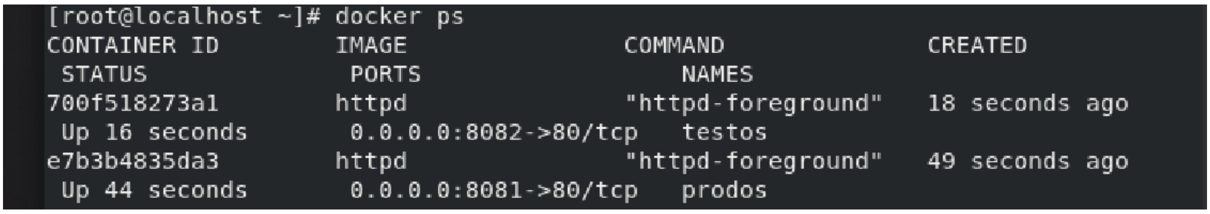
Job 1:- When dthe eveloppushesusthe h to master branch tJenkinskins will clthe one rethe po from master branch and deploy on httpd docker container which we will be used for productionenvironment . Let's name this container prodos

I am exposing this port to 8081. This job will automatically trigger and deploy our code into the webserver. To access the webpage we can the use coURLiner url by specifying the port number.

Job 2 :- When the Developer push to the dev branch or any new comis mit made by Jenkins then Jenkins will clonthe e from dev branch and deploy on another container. This container will be used by Quality Assurance Team to test all the features and quality applicationppation. Let's name this testsner testos

I am exposing this container to port 8082.

To see the container running



```
[root@localhost ~]# docker ps
CONTAINER ID        IMAGE               COMMAND             CREATED
 STATUS              PORTS                   NAMES
700f518273a1        httpd               "httpd-foreground"   18 seconds ago
 Up 16 seconds       0.0.0.0:8082->80/tcp   testos
e7b3b4835da3        httpd               "httpd-foreground"   49 seconds ago
 Up 44 seconds       0.0.0.0:8081->80/tcp   prodos
```

Job 3:- This job is for the Quality Assurance team. They check the feature addeda in testing environment. If this feature meets the requirement then this feature is ready to launa ch in Production environment.

For deploying in the production environment we need to methe rge dev branthe ch with master branch. Once dev is a merged with master branch a new cotoit is the added in masthe ter branch and job will run.

```
# sudo docker container rm -f prodos
```

Now, this setup is done.

After testing the website in a testing environment. Quality Assurance Team trigger this Job3.

Finally, this dev code is deployedinna production environmethatere the client can ace itss.

8. What is a Docker data center?

Docker Datacenter (DDC) is **the Containers as a Service (CaaS) platform for enterprise IT and application teams**. **to build, ship, and run, any application anywhere**. The latest DDC release includesseveralf features an

. What is Docker hub & uses?

Docker Hub provides the following major features: Repositories: Push and pull container images. Teams & Organizations: Manage access to private repositories of container images. Docker Official Images: Pull and use high-quality container images provided by Docker.

10. **What are the types of Docker networks**?

There are three common Docker network types – **bridge networks, used within a single host, overlay networks, for multi-host communication, and macvlan networks** which are used to connect Docker containers directly to host network interfaces.

11. How do you define the network in the Docker compose file?

12. **What are the basic parameters required in the Docker compose file?**

13. **What is Docker interlock**?

Interlock is **an application routing proxy service for Docker**. Fully integrates with Docker (Swarm, Services, Secrets, Configs) Enhanced configuration (context roots, TLS, zero downtime deploy, rollback) Support external load balancers (nginx, proxy, etc) via extensions.

14. What is overlay networking?

Overlay networking (aka SDN overlay) is a method of using software to create layers of network abstraction that can be used to run multiple .

Other examples of overlay network deployments include virtual private networks (VPNs), peer-to-peer (P2P) networks, content delivery networks (CDNs), voice over IP (VoIP) services such as Skype, and non-native software-defined networks.

15. How to communicate between 2 containers present in a separate network?

16. How to store the data present in the Docker container in the AWS?

17. If we define the Docker volume in the docker-compose file is it possible to share

data with the EFS, and NFS?

18. Difference between image and container

Image is created only once. Containers are created any number of times using an images. Images are immutable. Containers changes onlythe if old image is deleted and new is used to build the container

19. How to Run containers

Run in detached mode

Docker can run your container in detached mode or the background. To do this, we can **use the --detach or -d for short**. Docker will start your container the same as before but this time will "detach" from the container and return you to the terminal prompt

20. Why do we need to mention dual ports(8080:8080) in the docker run command

8080:80 refers that **in the container you are using port 80 and you are forwarding that port to the host machine's 8080 port**. So you are running Jenkins on port 80 inside your container wherever in scenario 2 you are running Jenkins on port 8080 inside the container and exposing it over the same portthe on host machine

the network ports required for a Docker Swarm to function correctly are:

- TCP port 2376 for secure Docker client communication. ...
- TCP port 2377 . ...
- TCP and UDP port 7946 for communication among nodes (container network discovery).
- UDP port 4789 for overlay network traffic (container ingress networking).

21. .Difference between Copy and Add

- ADD only adds files to the image file system, whereas COPY adds files as well as directoriesthe to imfile systemstem.
- ADD is used by a single docker host whereas COPY is used by docker swarm clustes .

COPY takes in an src and destination. It only lets you copy in a local or directory from your host (the machine-building the Docker image) into the Docker image itself. ADD lets you do that too, but it also supports 2 other sources. First, you can use a URL instead of a local file/directory

22. How to write a docker file to deploy a war file

war, which we need to deploy to the Tomcat server. To achieve our goal, we need to first create a Dockerfile.
...

3. Deploy WAR in Docker Container

1. 3.1. Create Dockerfile. ...
2. 3.2. Build the Docker Image. ...
3. 3.3. Run Docker Container. ...
4. 3.4. Verify the Setup.

Let's assume that we have a WAR file for our application, ROOT. war, which we need to deploy to the Tomcat server. To achieve our goal, we need to first create a Dockerfile.
...

i. We have a private repository and don't have base images .how can deploy a war file?

3. Deploy WAR in Docker Container

1. 3.1. Create Dockerfile. ...
2. 3.2. Build the Docker Image. ...
3. 3.3. Run Docker Container. ...
4. 3.4. Verify the Setup.

ii. Write a simple docker file to deploy a war file by using base images?

23. Difference between docker-compose and docker swarm

Docker Compose is used for configuring and starting multiple Docker containers on the same host–so you don't have to start each container separately. **Docker swarm is a container orchestration tool that allows you to run and connect containers on multiple hosts**.

24. why do we need to use docker-compose?

Docker Compose is a tool that was developed **to help define and share multi-container applications**. With Compose, we can create a YAML file to define the services and with a single command, can spin everything up or tear it all down

25. Tell me about Docker Network.

Docker networking is **primarily used to establish communication between Docker containers and the outside world via the host machine where the Docker daemon is running**. Docker supports different types of networks, each fit for certain use cases.

Linux Interview questions:

----

1. script/command to delete last word from every line in a file

```
awk '{gsub("[a-zA-Z0-9]*$", "");print}' <filename>
```

To replace last word with hello in every line

```
awk '{gsub("[a-zA-Z0-9]*", "hello");print}' <filename>
```

2. script/command to find the files with more than 1GB size

```
find <path for directory> -size +1G -type f
```

3. What is Swap Space?

Swap space is **a space on a hard disk that is a substitute for physical memory**. It is used as virtual memory which contains process memory images. Whenever our computer runs short of physical memory it uses its virtual memory and stores information in memory on a disk

4. What is the maximum length for a file name in Linux?

255 bytes

On Linux: The maximum length for a file name is **255 bytes**. The maximum combined length of both the file name and path name is 4096 bytes.

5. Which partition stores the system configuration files in the Linux system?

standard root partition

The standard root partition (indicated with a single forward slash, /) is about 100-500 MB and contains the system configuration files, most basic commands and server programs, system libraries, some temporary space ,and the home directory of the administrative user.

6. Which command is used to uncompress gzip files?

Gunzip is a command-line tool for decompressing Gzip files

7. What is the difference between soft and hard mounting points?

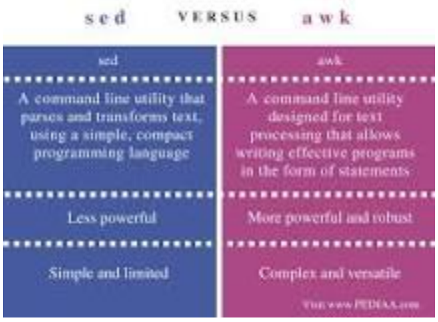
A hard mount is generally used for block resources like a local disk or SAN. A soft mount is usually used for network file protocols like NFS or CIFS. The advantage of a soft mount is that if your NFS server is unavailable, the kernel will time out the I/O operation after a pre-configured period

8. What are the file permissions in Linux?

(read)	read file content (cat)	read directory content (ls)
w (write)	change file content (vi)	create a file in directory (touch)
x (execute)	execute the file	enter the directory (cd)

9. more questions are from sed, find, and awk.

What is the difference between sed and awk in Linux?



sed VERSUS awk

sed	awk
A command line utility that parses and transforms text, using a simple, compact programming language	A command line utility designed for text processing that allows writing effective programs in the form of statements
Less powerful	More powerful and robust
Simple and limited	Complex and versatile

Visit www.PEDIAA.com

The main difference between sed and awk is that sed is a command utility that works with streams of characters for searching, filtering and text processing while awk is more powerful and robust than sed with sophisticated programming constructs such as if/else, whle, do/whi,le etc.

10. How to check Memory stats and CPU stats as a Linux admin?

1. free command. The free command is the most simple and easy-to-use command to check memory usage on linux. ...
2. 2. /proc/meminfo. The next way to check memory usage is to read the /proc/meminfo file. ...
3. vmstat. ...
4. top command. ...
5. htop.

Or

1. How To Check CPU Usage from Linux Command Line. top Command to View Linux CPU Load. mpstat Command to Display CPU Activity. sar Command to Show CPU Utilization. iostat Command for Average Usage.
2. Other Options to Monitor CPU Performance. Nmon Monitoring Tool. Graphical Utility Option.

How to Check CPU Usage

1. Start the Task Manager. Press the buttons Ctrl, Alt and Delete all at the same time. ...
2. Choose "Start Task Manager." This will open the Task Manager Program window.
3. Click the "Performance" tab. In this screen, the first box shows the percentage of CPU usage.

11. How to reduce or shrink the size of the LVM partition?

1. Step 1: First take a full backup of your filesystem.
2. Step 2:Start and force a filesystem check.
3. Step 3:Resize your filesystem before resize your Logical Volume.
4. Step 4: Reduce LVM size.
5. Step 5: Re-run resize2fs.

   What is LVM partitioning?
   LVM (**Logical Volume Management**) partitions provide a number of advantages over standard partitions. LVM partitions are formatted as physical volumes. One or more physical volumes are combined to form a volume group. Each volume group's total storage is then divided into one or more logical volumes.

12. How can you enhance the security of password files?

Two common techniques to protect a password file are- **hashed passwords as well as a salt value or password file access control**.

13. What is the difference between Cron and Anacron?

Cron runs the scheduled jobs at a very specific interval, but only if the system is running at that moment. However, Anacron runs the scheduled job even if the computer is off at that moment. It runs those missed jobs once you turn on the system

14. What command is used to check the number of files, disk space and each user's defined quota?

check disk space with df command

Display the quotas and disk use for all users on one or more file systems by using the **repquota command**.

Find the Number of Files in a Directory. We can use the **ls command along with the wc command** to count the number of files in a directory.

16. What is the name and path of the main system log?

Name of the main system log is **"messages" and path is /var/log/messages**

17. how to manage logical volumes?

Logical volume management (LVM) is **a form of storage virtualization that offers system administrators a more flexible approach to managing disk storage space than traditional partitioning**. This type of virtualization tool is located within the device-driver stack on the operating system.

18. Explain /proc filesystem?

The proc filesystem (procfs) is **a special filesystem in Unix-like operating systems that presents information about processes and other system information in a hierarchical file-like structure**, providing a more convenient and standardized method for dynamically accessing process data held in the kernel than traditional ..

19. What are the fields in the/etc/passwd file?

The /etc/passwd file is a colon-separated file that contains the following information:

- User name.
- Encrypted password.
- User ID number (UID)
- User's group ID number (GID)
- Full name of the user (GECOS)
- User home directory.
- Login shell.

20. How do you terminate an ongoing process?

1. (Optional) To terminate the process of another user, become superuser or assume an equivalent role.
2. Obtain the process ID of the process that you want to terminate. $ ps -fu user. ...

3. Terminate the process. $ kill [signal-number] pid. ...
4. Verify that the process has been terminated.

21. How can you know the execution time of a command?

The time command in Linux is used to determine the duration of execution of a command.

Check running process time using ps

1. etime option displays elapsed time since the process was started, in the form [[DD-]hh:]mm: ss. ...
2. PID --> ID of the running process.
3. STARTED --> The time the process was initially started.
4. ELAPSED --> Total running time of the process.
5. COMMAND --> Process executed command.

22. How can you append one file to another in Linux?

You can use the **cat command** to append data or text to a file. The cat command can also append binary data.

23. How you can run a Linux program in the background simultaneously when you start your Linux Server?

By using nohup. It will stop the process receiving the NOHUP signal and thus terminating it you log out of the program which was invoked with. & runs the process in the background.

AWS Interview Questions

1. What is AWS?

AWS (Amazon Web Services) is a comprehensive, evolving cloud computing platform provided by Amazon that includes a mixture of infrastructure as a service (IaaS), platform as a service (PaaS) and packaged software as a service (SaaS) offerings.

2. What is IAM and its purpose?

AWS Identity and Access Management (IAM) is **a web service that helps you securely control access to AWS resources**. You use IAM to control who is authenticated (signed in) and authorized (has permissions) to use resources.

3. What are the policies in IAM?

policy is an entity that, when attached to an identity or resource, defines their permissions. You can use the AWS Management Console, AWS CLI, or AWS API to create *customer managed policies* in IAM. Customer managed policies are standalone policies that you administer in your own AWS account. You can then attach the policies to identities (users, groups, and roles) in your AWS account.

A policy that is attached to an identity in IAM is known as an *identity-based policy*. Identity-based policies can include AWS managed policies, customer managed policies, and inline policies. AWS managed policies are created and managed by AWS. You can use them, but you can't manage them. An inline policy is one that you create and embed directly to an IAM group, user, or role. Inline policies can't be reused on other identities or managed outside of the identity where it exists. For more information, see Adding and removing IAM identity permissions.

4. How to set up MFA?

To configure and enable a virtual MFA device for use with your root user (console)

1. Sign in to the AWS Management Console.
2. On the right side of the navigation bar, choose your account name, and choose My Security Credentials. ...
3. Choose Activate MFA.
4. In the wizard, choose Virtual MFA device, and then choose Continue.

5. Two types of access for IAM users?

Access keys: **A combination of an access key ID and a secret access key**. You can assign two to a user at a time. These can be used to make programmatic calls to Amazon.

Cross-Account Access: granting permissions to users from other AWS account, whether you control those account or not. Identity Provider Access: granting permissions to users authenticated by a trusted external system

6. What is the EC2 service used for?

You can use Amazon EC2 to **launch as many or as few virtual servers as you need, configure security and networking, and manage storage**. Amazon EC2 enables you to scale up or down to handle changes in requirements or spikes in popularity, reducing your need to forecast traffic.

7. Name a few types of EC2 instances?

Here are different types of EC2 Instances:

- General Purpose Instances.
- Compute Optimized Instances.
- Memory-Optimized Instances.
- Accelerated Computing Instances.
- Storage Optimized Instances.

8. Diff between T2 series and C series on Ec2 instances?

T2 instances are **Burstable Performance Instances that provide a baseline level of CPU performance with the ability to burst above the baseline**. The baseline performance and ability to burst are governed by CPU Credits. T2 instances accumulate CPU Credits when they are idle, and consume CPU Credits when they are active.

T3 offers better value than T2, including extra CPU credits and better baseline performance, faster CPUs. The downside is the migration effort. The cost difference of the instance types between Windows and Linux is considerable and results in different optimal instance types: For Linux, T3 is priced lower than T2.

9. What are amis?

An Amazon Machine Image (AMI) is used **to create virtual servers (Amazon Elastic Compute Cloud or EC2 instances) in the Amazon Web Services (AWS) environment**.

Different types of instances can be launched from a single AMI to support the hardware of the host computer used for the instance.

10. What is AWS marketplace, what are community amis?

Community AMIs: Whenever you create an AMI, you can add permissions to it to make it public. In that case, it goes to "community AMIs". These are **AMIs that comes from AWS users, and are not verified by AWS**. Makerplace: this is a whole service at AWS, and all AMIs here are verified by AWS

11. How can you create your own AMI?

You can launch an instance from an existing AMI, customize the instance (for example, install software on the instance), and then save this updated configuration as a custom AMI. Instances launched from this new custom AMI include the customizations that you made when you created the AMI.

12. Enable termination protection option is for what?

Termination protection prevents an instance from accidental termination. By default, this option is disabled for EC2 instances. Enable this option **to protect your instance from any unintentional termination**

13. What is EBS?

Amazon Elastic Block Store (Amazon EBS) **provides block level storage volumes for use with EC2 instances**. EBS volumes behave like raw, unformatted block devices. You can mount these volumes as devices on your instances.

14. What are the different types of Volumes?

The three types that are now available include **Magnetic, Provisioned IOPS (SSD) and General Purpose (SSD) EBS volumes**. All three have their merits and offer similar functionalities, such as snapshot capabilities, though they differ largely in cost and performance.

15. Can we create volume in one zone and attach it to an instance in the other zone?

After you create a volume, **you can attach it to any EC2 instance in the same Availability Zone**

16. How to take backups of EBS volume?

You can create a snapshot manually from the console. **On the Amazon EC2 console, on the Elastic Block Store Volumes page, select the volume that you want to back up. Then on the Actions menu, choose Create Snapshot**.

17. How to restore lost/corrupted volumes from snapshots?

or example, follow these steps to restore a volume to an earlier point-in-time backup by using the console:

1. On the Amazon EC2 console, on the **Elastic Block Store** menu, choose **Snapshots**.
2. Search for the snapshot that you want to restore, and select it.
3. Choose **Actions**, and then choose **Create Volume**.
4. Create the new volume in the same Availability Zone as your EC2 instance.
5. On the Amazon EC2 console, select the instance.
6. In the instance details, make note of the device name that you want to replace in the **Root device** entry or **Block Devices** entries.

7. Attach the volume. The process differs for root volumes and non-root volumes.

   For root volumes:

   . Stop the EC2 instance.
   a. On the **EC2 Elastic Block Store Volumes** menu, select the root volume that you want to replace.
   b. Choose **Actions**, and then choose **Detach Volume**.
   c. On the **EC2 Elastic Block Store Volumes** menu, select the new volume.
   d. Choose **Actions**, and then choose **Attach Volume**.
   e. Select the instance that you want to attach the volume to, and use the same device name that you noted earlier.

   For non-root volumes:

   f. On the **EC2 Elastic Block Store Volumes** menu, select the root volume that you want to replace.
   g. Choose **Actions**, and then choose **Detach Volume**.
   h. Attach the new volume by choosing it on the **EC2 Elastic Block Store Volumes** menu and then choosing **Actions**, **Attach Volume**. Select the instance that you want to attach it to, and then select an available device name.
   i. Using the operating system for the instance, unmount the existing volume, and then mount the new volume in its place.

      In Linux, you can use the `umount` command. In Windows, you can use a logical volume manager (LVM) such as the Disk Management system utility.
   j. Detach any prior volumes that you may be replacing by choosing it on the **EC2 Elastic Block Store Volumes** menu and then choosing **Actions**, **Detach Volume**.

You can also use the AWS CLI in combination with operating system commands to automate these steps.

18. How to transfer volume from one zone to another zone?

How do I transfer EBS volume from one region to another?

Answer

1. Go to the volume where your EBS snapshot resides.
2. Select the EBS snapshot you want to copy to another region and then click on the Copy Snapshot button.
3. Put a name and description on the EBS snapshot you want to copy to another region and then select the region you want to copy it to.

following are the Steps involved in Migrating the EC2 instance from one AZ to another:

1. Login in AWS console.

2. Go to Compute service –> EC2.
3. Create an EC2 instance.
4. Go to the EBS –> Volumes –> Check the Additional information and match instance id of EC2 machine with EBS volume. ...
5. Select the EBS volume and go to action tab.

How do I transfer EBS volume from one region to another?

19. How to resize EBS volumes.

Expand the root volume. Then, extend the file system using the Amazon EC2 console (new console)

How do I transfer EBS volume from one region to another?

Answer

1. Go to the volume where your EBS snapshot resides.
2. Select the EBS snapshot you want to copy to another region and then click on the Copy Snapshot button.
3. Put a name and description on the EBS snapshot you want to copy to another region and then select the region you want to copy it to.

1. From the Amazon EC2 console, choose Instances from the navigation pane.
2. Select the instance that you want to expand. ...
3. Select the volume. ...
4. In the Size field, enter the Size. ...
5. Choose Modify, and then choose Yes.

20. Modify volume feature from AWS?

https://aws.amazon.com/blogs/aws/amazon-ebs-update-new-elastic-volumes-change-everything/

Open the Amazon EC2 console at https://console.aws.amazon.com/ec2/ . In the navigation pane, choose Volumes. Select the volume to modify and choose Actions, Modify volume. The Modify volume screen displays the volume ID and the volume's current configuration, including type, size, IOPS, and throughput.

21. What are security groups?

Security Group

- A security group is a virtual firewall which is controlling the traffic to your EC2 instances.
- When you first launch an EC2 instance, you can associate it with one or more security groups.
- A Security group is the first defence against hackers

22. Can we attach a Security group to multiple instances?

Single security groups can be applied to multiple instances, in the same way that you can apply a traditional security policy to multiple firewalls.

23. What is the diff between inbound and outbound rules in SG?

Inbound rules control the incoming traffic to your instance, and outbound rules control the outgoing traffic from your instance.

24. What are Elastic IPs and how is it different from normal Public IP assigned to an EC2 instance?

Elastic IP is used when you are working on long time project and configuration of IP sometime consumes more time. Public IP is used when you are working on small projects and running 2-3 servers. Here in this situation you make use of IP for short time

25. What are Key pairs?

A key pair is **a combination of a public key that is used to encrypt data and a private key that is used to decrypt data**.

26. What is ELB?

Elastic Load Balancing (ELB) is **a load-balancing service for Amazon Web Services (AWS) deployments**. ELB automatically distributes incoming application traffic and scales resources to meet traffic demands. ELB helps an IT team adjust capacity according to incoming application and network traffic.

27. What are target groups?

A target group **tells a load balancer where to direct traffic to : EC2 instances, fixed IP addresses; or AWS Lambda functions, amongst others**. When creating a load balancer, you create one or more listeners and configure listener rules to direct the traffic to one target group.

28. What are health check settings in the target group?

Your Application Load Balancer periodically sends requests to its registered targets to test their status. These tests are called health checks. Each load balancer node routes requests only to the healthy targets in the enabled Availability Zones for the load balancer.

29. Describe advanced health check settings from Target groups?

These tests are called health checks. Each load balancer node routes requests only to the healthy targets in the enabled Availability Zones for the load balancer. **Each load balancer node checks the health of each target, using the health check settings for the target groups with which the target is registered**

30. Difference between internet-facing and internal ELB?

The DNS name of an internet-facing load balancer is publicly resolvable to the public IP addresses of the nodes. Therefore, internet-facing load balancers can route requests from clients over the internet. **The nodes of an internal load balancer have only private IP addresses**.

31. What is VPC?

ogically isolated from other virtual networks in the AWS Cloud. You can launch your AWS resources, such as Amazon EC2 instances, into your VPC.

32. What is the default VPC?

default VPC is **a logically isolated virtual network in the AWS cloud that is automatically created for your AWS account the first time you provision Amazon EC2 resources**. When you launch an instance without specifying a subnet-ID, your instance will be launched in your default VPC.

33. Can we create default VPC?

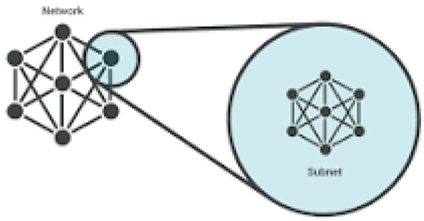
Amazon Virtual Private Cloud (VPC) now allows customers to create a new default VPC directly from the console or by using the CLI. With this release, customers no longer need to contact AWS support if the default VPC has been deleted, as they can create a new default VPC by using this self-service feature.

34. What is the DNS hostnames option used for in VPC?

Domain Name System (DNS) is a standard by which names used on the internet are resolved to their corresponding IP addresses. A DNS hostname uniquely names a computer and consists of a host name and a domain name. **DNS servers resolve DNS hostnames to their corresponding IP addresses**.

35. What are Subnets in VPC?

What are subnets in cloud?



A subnet, or subnetwork, is **a network inside a network**. Subnets make networks more efficient. Through subnetting, network traffic can travel a shorter distance without passing through unnecessary routers to reach its destination.

Why do we create subnets in VPC?

Creating multiple Subnets **allows you to create logical network divisions between your resources**. By doing so, you could have a Subnet for database instances, another for application servers, and another for web infrastructure. By splitting up your Subnets this way, helps to enforce a greater level of security.

36. What is the difference between the Private and Public subnet?

The instances in the public subnet can send outbound traffic directly to the internet, whereas the instances in the private subnet can't. Instead, the instances in the private subnet can access the internet by using a network address translation (NAT) gateway that resides in the public subnet.

37. How can you design a Highly available network in VPC?

reate 2 public and 2 private subnets within the VPC. Configure a Route Table within Your VPC Named PubRT. Configure a route table within your VPC named PubRT that contains the necessary routes for public connectivity. Create the Following Security Groups: PublicSG and PrivateSG.

38. What are Internet Gateways?

An internet gateway is **a horizontally scaled, redundant, and highly available VPC component that allows communication between your VPC and the internet**.

39. What are Nat gateways?

NAT Gateway is **a highly available AWS managed service that makes it easy to connect to the Internet from instances within a private subnet in an Amazon Virtual Private Cloud (Amazon VPC)**. Previously, you needed to launch a NAT instance to enable NAT for instances in a private subnet.

40. What are Route tables?

The route table **contains existing routes with targets other than a network interface, Gateway Load Balancer endpoint, or the default local route**. The route table contains existing routes to CIDR blocks outside of the ranges in your VPC. Route propagation is enabled for the route table.

41. What is NACL?

What is an AWS NACL? In AWS, a network ACL (or NACL) **controls traffic to or from a subnet according to a set of inbound and outbound rules**. This means it represents network level security.

42. What is VPC Peering?

A VPC peering connection is **a networking connection between two VPCs that enables you to route traffic between them using private IPv4 addresses or IPv6 addresses**. Instances in either VPC can communicate with each other as if they are within the same network.

43. What is RDS?

Amazon Relational Database Service (RDS) is **a managed SQL database service provided by Amazon Web Services (AWS)**. Amazon RDS supports an array of database engines to store and organize data. It also helps with relational database management tasks, such as data migration, backup, recovery and patching.

44. What are different DB software(Engines) supported by RDS?

Choose from seven popular engines — **Amazon Aurora with MySQL compatibility, Amazon Aurora with PostgreSQL compatibility, MySQL, MariaDB, PostgreSQL, Oracle, and SQL Server —** and deploy on-premises with Amazon RDS on AWS Outposts

45. Diffrent DB instance types?

Amazon RDS supports three types of instance classes: **Standard, Memory Optimized, and Burstable Performance**

46. What are MutiAZ RDS instances?

n an Amazon RDS Multi-AZ deployment, **Amazon RDS automatically creates a primary database (DB) instance and synchronously replicates the data to an instance in a different AZ**. When it detects a failure, Amazon RDS automatically fails over to a standby instance without manual intervention.

47. Diffrent storage types in RDS?

DS provides three types of storage:

- General-purpose solid-state drive (SSD). Amazon recommends this storage as the default choice.
- Provisioned input-output operations per second (IOPS). SSD storage for I/O-intensive workloads.
- Magnetic. A lower cost option.

48. What is a backup retention policy?

A backup retention policy is **an internal organizational rule that determines what data the organization keeps, where it keeps the data and how long it keeps the data**. Retention policies may indicate the types of backup that are acceptable.

backup retention policy in AWS?

Amazon RDS retains backups of a DB Instance for a limited, user-specified period of time called the retention period, which by default is 7 days but can be set to up to 35 days. You can initiate a point-in-time restore and specify any second during your retention period, up to the Latest Restorable Time.

49. What is autoscaling? What triggers have you used for scaling?

The Auto Scaling group in your Elastic Beanstalk environment uses two Amazon CloudWatch alarms to trigger scaling operations. The default triggers scale **when the average outbound network traffic from each instance is higher than 6 MB or lower than 2 MB over a period of five minutes**.

50 . What is s3cmd used for?

What is S3cmd. S3cmd (s3cmd) is a free command line tool and client for **uploading, retrieving and managing data in Amazon S3 and other cloud storage service providers that use the S3 protocol, such as Google Cloud Storage or DreamHost DreamObjects**.

51. What is AWS CLI used for? Give any real-time scenario where you used AWS CLI?

The AWS Command Line Interface (CLI) is a unified tool **to manage your AWS services**. With just one tool to download and configure, you can control multiple AWS services from the command line and automate them through scripts.

52. How can we mount s3bucket in a Linux instance as a mount point? Any real-time use case for it?

Follow the below steps to mount your S3 bucket to your Linux Instance:
Step 1: Download latest s3fs package and extract:

```
wget https://storage.googleapis.com/google-code-archive-downloads/v2/code.google.com/s3fs/s3fs-1.74.tar.gz
tar -xvzf s3fs-1.74.tar.gz
```

Step 2: Update OS and install dependencies as mentioned in above pre-req.

Step 3: Now change to extracted directory, compile and install s3fs source code.

```
cd s3fs-1.74

./configure –prefix=/usr

make

make install
```

Step 4: Use below command to check where s3fs command is placed in O.S. It will also confirm whether installation is ok:

```
which s3fs
```

Step 5: Get IAM user Access and secret key which have appropriate permissions (e.g. S3 Full access), You can get the same from AWS IAM console

Step 6: Create a new file in /etc with the name passwd-s3fs and Paste the access key and secret key in the below format and change the permission for the file:
echo "AccessKey:SecretKey" > /etc/passwd-s3fs
chmod 640 /etc/passwd-s3fs

Note: Replace AcessKey and SecretKey with original keys.

Step 7: Now create a directory and mount S3bucket in it. Here, Provide your S3 bucket name in place of "your_bucketname"

53. Please explain the procedure for setting up an alarm with Cloud Watch monitoring?

Open the Amazon EC2 console at https://console.aws.amazon.com/ec2/ . In the navigation pane, choose Instances. Select the instance and choose Actions, Monitor and troubleshoot, Manage CloudWatch alarms. On the Manage CloudWatch alarms detail page, under Add or edit alarm, select Create an alarm.

54. What is AWS Beanstalk? Name any platform you used in beanstalk and the code deployment procedure for it?

AWS Elastic Beanstalk is an easy-to-use service for **deploying and scaling web applications and services** developed with Java, . NET, PHP, Node. js, Python, Ruby, Go, and Docker on familiar servers such as Apache, Nginx, Passenger, and IIS.

Elastic Beanstalk provides platforms for **programming languages (Go, Java, Node. js, PHP, Python, Ruby), application servers (Tomcat, Passenger, Puma), and Docker containers**. Some platforms have multiple concurrently-supported versions.

To deploy a new application version to an Elastic Beanstalk environment

1. Open the Elastic Beanstalk console, and in the **Regions** list, select your AWS Region.
2. In the navigation pane, choose **Environments**, and then choose the name of your environment from the list.

   Note

   If you have many environments, use the search bar to filter the environment list.
3. Choose **Upload and deploy**.
4. Use the on-screen form to upload the application source bundle.
5. Choose **Deploy**.

55. Name a few customization options available with Beanstalk?

AWS Elastic Beanstalk can now be customized and configured via YAML configuration files. You can use configuration files to **download and install packages, download and extract archives, create files, create users/groups, run commands, start and stop services, and define container settings**

56. What is Launch configuration in Autoscaling?

A launch configuration is **an instance configuration template that an Auto Scaling group uses to launch EC2 instances**. When you create a launch configuration, you specify information for the instances.

57. What is Route 53 used for, name a few features of route 53?

Amazon Route 53 is **a highly available and scalable cloud Domain Name System (DNS) web service**. It is designed to give developers and businesses an extremely reliable and cost effective way to route end users to Internet applications by translating names like www.example.com into the numeric IP addresses like 192.0.

58. Difference between AWS Region and Zone?

Each Region is a separate geographic area. Availability Zones are multiple, isolated locations within each Region. Local Zones provide you the ability to place resources, such as compute and storage, in multiple locations closer to your end users.

59. How will you identify which one is public subnet & private subnet in VPC

There's no definite way to identify public and private subnets without looking at their routing tables: a public subnet will route to an Internet Gateway, while a private subnet won't.

From the AWS docs: If a subnet is associated with a route table that has a route to an internet gateway, it's known as a public subnet. If a subnet is associated with a route table that does not have a route to an internet gateway, it's known as a private su

60. Is Amazon EBS storage is NAC/SAN storage?

EBS is on a SAN. Its not a NAS. Its on a Storage Array that gives you a higher throughput, and practically should be within the primary physical infra where your instance is running.

61. What are services available to connect your premise data center to the AWS cloud?

AWS Direct Connect enables you to securely connect your AWS environment to your on-premises data center or office location over a standard 1 gigabit or 10 gigabit Ethernet fiber-optic connection.

62. Can AWS EC2 access the s3 bucket without an access key?

You can access an S3 bucket privately without authentication when you access the bucket from an Amazon Virtual Private Cloud (Amazon VPC). However, make sure that the VPC endpoint used points to Amazon S3.

63. I want to deploy containers but have storage persistent problems in AWS and in

Docker?

64. How are you using AWS in your project/company?

65. Difference between IAM Roles & Policies?

IAM Roles manage who has access to your AWS resources, whereas IAM policies control their permissions. A Role with no Policy attached to it won't have to access any AWS resources

66. Describe your AWS Architecture in your company?

67. What is RDS what is the work you have done in RDS?

amazon Relational Database Service (RDS) is a managed SQL database service provided by Amazon Web Services (AWS). Amazon RDS supports an array of database engines to store and organize data. It also helps with relational database management tasks, such as **data migration, backup, recovery and patching**.

68. What is cloud formation?

CloudFormation is **a method of provisioning AWS infrastructure using code**. It allows you to model a collection of related resources, both AWS and third party, to provision them quickly and consistently. AWS CloudFormation also provides you with a mechanism to manage the resources through their lifecycle

69. What is CDN?

A content delivery network (**CDN**) refers to a geographically distributed group of servers that work together to provide fast delivery of Internet content.

70. I have pdf files in an instance and they are eating away the space. I need these files,what is solution do you provide for accessing these files?

To resolve this issue, follow these steps:

1. Confirm that the DB instance status is STORAGE_FULL.
2. Add more storage space to the instance.
3. Increase the allocated storage property of your DB instance. If the DB instance is in a STORAGE_FULL state, the instance accepts only allocated storage modifications.

71. What is route 53? How do you configure it? And link it to ELB?

ELB distributes traffic among Multiple Availability Zone but not to multiple Regions. Route53 can distribute traffic among multiple Regions. In short, ELBs are intended to load balance across EC2 instances in a single region whereas DNS load-balancing (Route53) is intended to help balance traffic across regions.

72. What are private subnets? What is it needs?

Instances in the private subnet are **back-end servers that don't need to accept incoming traffic from the internet and therefore do not have public IP addresses**; however, they can send requests to the internet using the NAT gateway (see the next bullet).

73. What is the need for having your application or instances in private subnets when you

have the public domain through route 53?

A private hosted zone is a container that **holds information about how you want Amazon Route 53 to respond to DNS queries for a domain and its subdomains within one or more VPCs that you create with the Amazon VPC service**

74. Why do we need to use Glacier?

S3 Glacier **enables customers to offload the administrative burdens of operating and scaling storage to AWS**, so they don't have to worry about capacity planning, hardware provisioning, data replication, hardware failure detection and recovery, or time-consuming hardware migrations.

75. Tell me S3 reduced redundancy vs infrequent access?

Reduced Redundancy Storage (RRS) is **an Amazon S3 storage option that enables customers to store noncritical, reproducible data at lower levels of redundancy than Amazon S3's standard storage**.

Amazon S3 Standard - Infrequent Access (Standard - IA) is **an Amazon S3 storage class for data that is accessed less frequently, but requires rapid access when needed**. Standard - IA offers the high durability, throughput, and low latency of Amazon S3 Standard, with a low per GB storage price and per GB retrieval fee.

76. Tell me S3 Bucket level permissions?

By default, all Amazon S3 buckets and objects are private. Only the resource owner which is the AWS account that created the bucket can access that bucket. The resource owner can, however, choose to grant access permissions to other resources and users.

Git Interview Questions

----

1. What is a Version control system or Source code manager?

2. Benefits of VCS or SCM?

3. What is Git and What is a repository in GIT?

4. Difference between Git and SVN?

5. Two types of git authentication?

Git supports two types of remotes: **SSH and HTTPS**. These two use completely distinct authentication systems. For HTTPS remotes, git authenticates with a username + password.

Git supports two types of remotes: **SSH and HTTPS**. These two use completely distinct authentication systems. For HTTPS remotes, git authenticates with a username + password. With GitHub, instead of a password, you can also use a Personal Access Token (PAT).

6. Branches in Git and its purpose?

In Git, branches are a part of your everyday development process. Git branches are effectively **a pointer to a snapshot of your changes**. When you want to add a new feature or fix a bug—no matter how big or how small—you spawn a new branch to encapsulate your changes.

What is the common branching pattern in GIT?

Git Flow Branch Strategy

The main idea behind the Git flow branching strategy is to isolate your work into different types of branches. There are five different branch types in total:

- Main
- Develop
- Feature
- Release
- Hotfix

The two primary branches in Git flow are *main* and *develop*. There are three types of supporting branches with different intended purposes: *feature*, *release*, and *hotfix*.

Git Flow: Pros & Cons

The Git flow branching strategy comes with many benefits, but does introduce a few challenges.

The Benefits of Git Flow:

1. The various types of branches make it easy and intuitive to organize your work.
2. The systematic development process allows for efficient testing.
3. The use of release branches allows you to easily and continuously support multiple versions of production code.

The Challenges of Git Flow:

1. Depending on the complexity of the product, the Git flow model could overcomplicate and slow the development process and release cycle.
2. Because of the long development cycle, Git flow is historically not able to support Continuous Delivery or Continuous Integration.

What is a branching strategy?

A “branching strategy” refers to the strategy a software development team employs when writing, merging, and shipping code in the context of a version control system like Git. Software developers working as a team on the same codebase must share their changes with each other. But how can they do this efficiently while avoiding malfunctions in their application? The goal of any branching strategy is to solve that problem and to enable teams to work together on the same source code without trampling on each other. A branching strategy defines how a team uses branches to achieve this level of concurrent development.

This article will first review the benefits and shortcomings of several common Git branching strategies. Then, we’ll compare those to trunk-based development to learn how the latter solves those shortcomings and enables modern software delivery practices through feature flag management.

7. What is Pull requests?

A pull request **occurs when a developer asks for changes committed to an external repository to be considered for inclusion in a project's main repository**. It is important to note that “pull requests” are a workflow method, and are not a feature of the version control system itself

8. How to install git in Linux & Windows?

Install Git on Linux

1. From your shell, install Git using apt-get: $ sudo apt-get update $ sudo apt-get install git.
2. Verify the installation was successful by typing git --version : $ git --version git version 2.9.2.

3. Configure your Git username and email using the following commands, replacing Emma's name with your own.

9. How to set up a repository through the command line?

Create a repository

1. In the upper-right corner of any page, use the drop-down menu, and select New repository.
2. Type a short, memorable name for your repository. ...
3. Optionally, add a description of your repository. ...
4. Choose a repository visibility. ...
5. Select Initialize this repository with a README.
6. Click Create repository.

Of course, if you're fine with that, you should just use the standard git init or git remote add method, and set it up the normal way. But, **Github does have a command line tool that can be used to easily create repos with a single command**.

10. How to set up a repository in GitHub and clone it?

Connect it to github

1. Go to github.
2. Log in to your account.
3. Click the new repository button in the top-right. You'll have an option there to initialize the repository with a README file, but I don't.
4. Click the “Create repository” button.

Clone Your Github Repository

1. Open Git Bash. If Git is not already installed, it is super simple. ...
2. Go to the current directory where you want the cloned directory to be added. ...
3. Go to the page of the repository that you want to clone.
4. Click on “Clone or download” and copy the URL.

11. What is the git clone command used for?

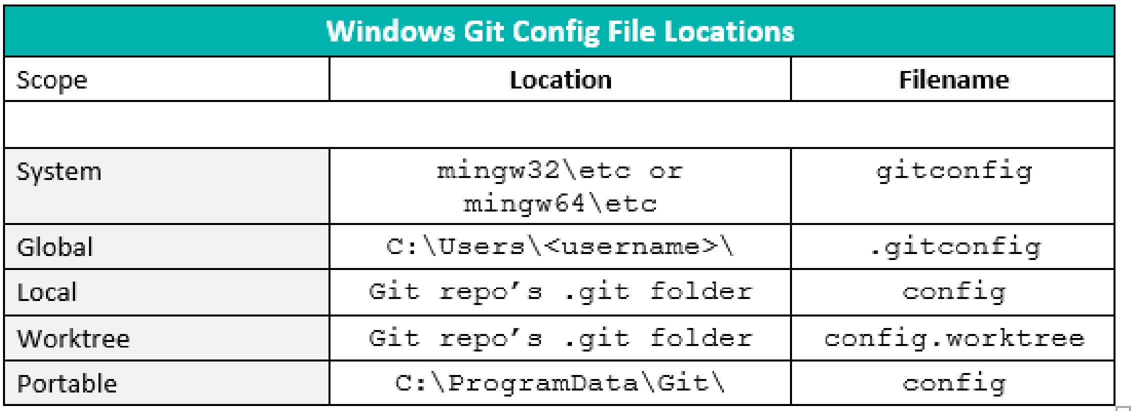
git clone is primarily used **to point to an existing repo and make a clone or copy of that repo at in a new directory, at another location**. The original repository can be located on the local filesystem or on remote machine accessible supported protocols. The git clone command copies an existing Git repository.

12. What is the git config command used for?

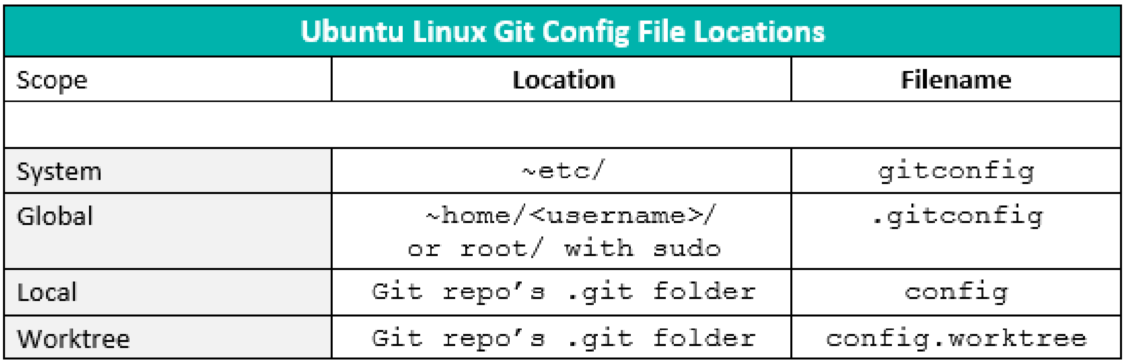
The git config command is a convenience function that is used **to set Git configuration values on a global or local project level**. These configuration levels correspond to . gitconfig text files. Executing git config will modify a configuration text file.

13. Git config data is stored in what location?

<tl;dr>



Windows Git Config File Locations		
Scope	Location	Filename
System	mingw32\etc or mingw64\etc	gitconfig
Global	C:\Users\<username>\	.gitconfig
Local	Git repo's .git folder	config
Worktree	Git repo's .git folder	config.worktree
Portable	C:\ProgramData\Git\	config



Ubuntu Linux Git Config File Locations		
Scope	Location	Filename
System	~etc/	gitconfig
Global	~home/<username>/ or root/ with sudo	.gitconfig
Local	Git repo's .git folder	config
Worktree	Git repo's .git folder	config.worktree

</tl;dr>

14. Git config global and local files?

Global Git config controls settings for the currently logged in user and all his repositories. Local **Git config controls settings for a specific repository**.

15. Content of git config file?

The Git configuration file contains **a number of variables that affect the Git commands' behavior**. The . git/config file in each repository is used to store the configuration for that repository, and $HOME/. gitconfig is used to store a per-user configuration as fallback values for the .

16. Git add command's purpose?

The git add command **adds a change in the working directory to the staging area**. It tells Git that you want to include updates to a particular file in the next commit.

17. How to remove/rename files in the local git repo?

Simply **git rm old or even git add -A** and it will realize that it is a rename. Git will see the delete plus the add with same content as a rename. You don't need to undo, unstage, use git mv etc

18. Does git commit the command's purpose?

The git commit command is one of the core primary functions of Git. Prior use of the git add command is required to select the changes that will be staged for the next commit. Then git commit is used to create a snapshot of the staged changes along a timeline of a Git projects history.

19. How to sync local git repo data with GitHub?

Syncing Central Repo with Local Repo

1. git pull upstream master - pull down any changes and sync the local repo with the central repo.
2. make changes, git add and git commit.
3. git push origin master - push your changes up to your fork.
4. Repeat.

20. Does git fetch?

The git fetch command **downloads objects to the local machine without overwriting existing local code in the current branch**. The command pulls a record of remote repository changes, allowing insight into progress history before adjustments. Read on to learn how to use the git fetch command through hands-on exampl

21. Does git merge?

Git can automatically merge commits unless there are changes that conflict in both commit sequences.

22. Does git pull?

git pull is a Git command used to update the local version of a repository from a remote. It is one of the four commands that prompts network interaction by Git. By default, git pull does two things. **Updates the remote tracking branches for all other branches**

23. How to change branches in the local git repo?

To create a new branch in Git, you use the **git checkout command and pass the -b flag with a name**. This will create a new branch off of the current branch. The new branch's history will start at the current place of the branch you "branched off of.

24. What is the difference between 'git remote' and 'git clone?

git remote is used to refer to a remote repository or your central repository. git clone is used to copy or clone a different repository.

25. git status?

The git status command **displays the state of the working directory and the staging area**. It lets you see which changes have been staged, which haven't, and which files aren't being tracked by Git. Status output does not show you any information regarding the committed project history.

26. What is the function of 'git rm'?

The primary function of git rm is to **remove tracked files from the Git index**. Additionally, git rm can be used to remove files from both the staging index and the working directory. There is no option to remove a file from only the working directory.

27. What is the function of 'git checkout' in git?

The git checkout command **lets you navigate between the branches created by git branch** . Checking out a branch updates the files in the working directory to match the version stored in that branch, and it tells Git to record all new commits on that branch

28. What is the use of 'git log'?

The git log command **displays all of the commits in a repository's history**. By default, the command displays each commit's: Secure Hash Algorithm (SHA) author

29. Explain what is commit message is?

A commit in GitHub is described as a saved change. A commit message **explains what change you made to your project**.

30. How to set up Github ssh authentication?

How to Add an SSH Key to your Github Account

1. Log into your GitHub account.
2. Click your avatar and choose Settings.
3. Select SSH and GPG keys.
4. Click New SSH key.
5. Enter a title in the field.
6. Paste your public key into the Key field.
7. Click Add SSH key.

31. What is git webhooks?

Webhooks **allow you to build or set up integrations, such as GitHub Apps or OAuth Apps, which subscribe to certain events on GitHub.com**. When one of those events is triggered, we'll send a HTTP POST payload to the webhook's configured URL.

33. Tell me the branching strategies you followed for your project?

A "branching strategy" refers to **the strategy a software development team employs when writing, merging, and shipping code in the context of a version control system like Git**. Software developers working as a team on the same codebase must share their changes with each othe

34. What is git rebase?

The git rebase command allows **you to easily change a series of commits, modifying the history of your repository**. You can reorder, edit, or squash commits together. Typically, you would use git rebase to: Edit previous commit messages.

35. What is git stash?

Git stash

- Definition ¶. The git stash command shelves changes you have made to your working copy so you can do another work, and then come back and re-apply them.
- Stashing your work ¶. ...
- Re-applying your stashed changes ¶. ...
- Stashing untracked or ignored files ¶. ...
- Multiple Stashes ¶.

36. What is the difference between git pull and git fetch?

git fetch is the command that tells your local git to retrieve the latest meta-data info from the original (yet doesn't do any file transferring. It's more like just checking to see if there are any changes available). git pull on the other hand does that AND brings (copy) those changes from the remote repository.

37. Difference between git and svn?

Below is a table of differences between GIT and SVN:

GIT	SVN
Git is open source distributed vice control system developed by Linus Torvalds in 2005. It emphasis on speed and data integrity	Apache Subversion is an open source software version and revision control system under Apache license.
Git has a Distributed Model.	SVN has a Centralized Model.
In git every user has their own copy of code on their local like their own branch.	In SVN there is central repository has working copy that also make changes and committed in central repository.
In git we do not required any Network to perform git operation.	In SVN we required Network for runs the SVN operation.
Git is more difficult to learn. It has more concepts and commands.	SVN is much easier to learn as compared to git.
Git deals with large number of files like binary files that change quickly that why it become slow.	SVN control the large number of binary files easily.
In git we create only .git directory.	In SVN we create .svn directory in each folder.
It does not not have good UI as compared to SVN.	SVN has simple and better user interface .
Features of GIT: • Distributed System. • Branching. • Compatibility. • Non-linear Development. • Lightweight. • Open source.	Features of SVN: • Directories are versioned • Copying, deleting, and renaming. • Free-form versioned metadata . • Atomic commits. • Branching and tagging. • Merge tracking. • File locking.

Here are some .git directory structures used in GIT:

- **HEAD/:** A pointer structure used in git.
- **Config/:** Contains all configuration preferences.
- **description/:** Description of your project.
- **index/:** It is used as a staging area between working directory.
- **object/:** All the data are stored here.
- **logs/:** Keeps record to change that are made.

38. Advantages of git compare with svn?

39. Explain about staging area in git?

The staging area is like a rough draft space, **it's where you can git add the version of a file or multiple files that you want to save in your next commit**

40. How do you check after git merge …where merging is correctly or not?

You can **use the diff-tree command with the -c flag**. This command shows you what files have changed in the merge commit. I got the -c flag's description from Git-Scm: This flag changes the way a merge commit is displayed (which means it is useful only when the command is given one , or --stdin).

41. How to create a branch and delete a branch?

1. create a branch and check it out.
2. let someone else delete it and create a new branch with the same name.
3. now do git branch -D <branch> and git checkout -b <branch> --track origin/<branch>
4. on a git pull you get ! [rejected] <branch> -> origin/<branch> (non-fast-forward)

42. Do you get any conflicts while merging..how can u resolve conflicts?

Merge conflicts **occur when competing changes are made to the same line of a file, or when one person edits a file and another person deletes the same file**. For more information, see "About merge conflicts."

43. How can u implement authentication and authorization in git?

Authenticating to Remote Git Repositories

1. Using a personal authentication token or password.
2. Using an SSH key.
3. Using your GitHub password with 2-factor authentication.

From your repository within GitHub, navigate to the settings screen from the list menu on the right side of the browser. From the settings go to the collaborators section using the link on the left side of the screen. Add your friend to the list of collaborators and they will be able to push to the repo.

44. What is the difference between git rebase and git merge?

“rebase reapplies commits on top of another base branch”, whereas “merge joins two or more development histories together”. In other words, the key difference between merge and rebase is that **while merge preserves history as it happened, rebase rewrites it**.

45. How to merge the code?

46. Pull request in git hub - ---git pull

47. Forking a repository in git hub?

Forking a repository **allows you to freely experiment with changes without affecting the original project**. Most commonly, forks are used to either propose changes to someone else's project or to use someone else's project as a starting point for your own idea.

48. Diff between svn and git ---svn is centralized version control and git is Distributed Version

Control.?

49. What is forking in git repositories?

Forking is **a git clone operation executed on a server copy of a projects repo**. A Forking Workflow is often used in conjunction with a Git hosting service like Bitbucket. A high-level example of a Forking Workflow is: You want to contribute to an open source library hosted at bitbucket.org/userA/open-project.

50. explain about soft and hard reset and their difference?

This reset is simply power cycling the cell phone turning it off and then back on. The Soft Reset does not cause any loss of data on the phone. The Hard Reset: Usually a last resort and the last reset to try when other types of resets have been attempted and have been unsuccessful in resolving the issue.

51. What kind of branching and merging do you work on GIT.

52. with this kind of environment and the git release branching, does it work in Agile methodology?

53. What is tagging in GIT and what is the diff between feature branching and tagging?

List of Cloudburn Interviews Questions:

1. Tell me about u r self briefly

2. Roles and Responsibilities

3. How to create a virtual host in apache

1. Step 1 — Create a conf file. Copy 000-default.com.conf to create a new file in /etc/apache2/sites-available : $ cd /etc/apache2/sites-available. ...
2. Step 2 — Modify the new conf file. In the example.com.conf : ...
3. Step 3 — Enabling a virtual host. ...
4. Step 4— Enabling SSL. ...
5. Step 5— Restart apache.

4. Versions(like Ansible, git, MySQL,)

Ansible 2.9

The current latest stable version of MySQL is **8.0**.

The current source code release is **version 2.35**.

The current LTS version announced on the Jenkins download page is **2.46. 3**.

Docker Engine 19.03 release notes.

Kubernetes 1.20,

The latest version of Nagios Core is **4.4. 7**

What is latest Grafana version?

8.4.

5. Write a playbook for installation of apache in ubuntu and centos

6. Document root of apache and tomcat

The DocumentRoot is **the top-level directory in the document tree visible from the web** and this directive sets the directory in the configuration from which Apache2 or HTTPD looks for and serves web files from the requested URL to the document root.

7. what is the configuration file in tomcat

The **server. xml** file is Tomcat's main configuration file, and is responsible for specifying Tomcat's initial configuration on startup as well as defining the way and order in which Tomcat boots and builds.

8. Defaults port numbers(apache, tomcat, MySQL)

Map **TCP port 80** in the container to port 8080 on the Docker host

In a typical Kubernetes cluster, the API serves on **port 443**, protected by TLS.

The default HTTP port that Grafana listens to is **3000** unless you have configured a different port.

Port 8080 exposes the web interface and port 50000 gives you **access to a remote Java (JIRA) API**.

Apache Tomcat will listen for requests on **port 8080**.

port 80

By default, Apache web server is instructed to listen for incoming connection and bind on **port 80**. If you opt for the TLS configuration, the server will listen for secure connections on port 443.
Port 3306 is the default port used for the MySQL protocol

You may remember the most common ones like HTTP, FTP, SSH but if you are working on various technology stacks then it's difficult to remember all of them.

Here I have listed the default port numbers of various applications to help you in the real world.

Application/Web Servers

Name	Port Number
Tomcat Startup	8080
Tomcat Startup (SSL)	8443
Tomcat Shutdown	8005
Tomcat AJP Connector	8009
GlassFish HTTP	8080
GlassFish HTTPS	8181
GlassFish Admin Server	4848
Jetty	8080
Jonas Admin Console	9000
IHS Administration	8008
JBoss Admin Console	8080
WildFly Admin Console	9990
WebLogic Admin Console	7001
WAS Admin Console (SSL)	9043
WAS Admin Console	9060
WAS JVM HTTP	9080 (first one only)
WAS JVM HTTPS	9443 (first one only)
Alfresco Explorer/Share	8080
Apache Derby Network Server	1527

Name	Port Number
OHS	7777
OHS (SSL)	4443
Jenkins	8080
Administrative server	4848
HTTP	8080
HTTPS	8181
IIPO	3700
IIOP_SSL	3820
IIOP_MUTUALAUTH and mutual authentication	3920
JMX_ADMIN	8686

Well-Known Common Protocols

Name	Port Number
FTP	20 and 21
HTTP	80
HTTPS	443
LDAP	389
LDAP (SSL)	636
SNMP	161
SSH	22
Telnet	23
SMTP	25
Microsoft RDP	3389
DNS Service	53
NNTP	119
IMAP	143
IMAP (SSL)	993
DNS	53
DHCP server	67
DHCP client	68
TFTP	69
SNMPTRAP	162
POP	110
NTP	123
Netstat	15
ARPA	42
Windows Internet Name Service	42
WHOIS	43
TACACS	49
Kerbos	88
SFTP	115
Network News Transfer Protocol	119
VMNET	175
BGP	179
IMAP	220
Border Gateway Multicast Protocol	264
POP3	995
Telnet	992

Database/Datastore

Name	Port Number
DB2	50000
Redis Server	6379
Oracle Listener	1521
mongoDB	27017
MySQL	3306
MS SQL	1433
Memcached	11211
MariaDB	3306
SQL Service	156

Messaging/Transfer

Name	Port Number
MQ Listener	1414
IBM Connect:Direct	1364
RabbitMQ Web UI	15672
Tibco RV Daemon	7474
GoToMyPC	8200

Misc

Name	Port Number
Syslog	514 (UDP)

9. How to connect client and server(Windows to Linux)

1. Download PuTTY. Use the following steps to download and open PuTTY: ...
2. Configure your connection. Use the following steps to configure your connection: ...
3. Accept the key. ...
4. Enter your username and password. ...
5. Change your root passwords.

10. What is SSH

SSH or Secure Shell is **a network communication protocol that enables two computers to communicate (c.f http or hypertext transfer protocol, which is the protocol used to transfer hypertext such as web pages) and share data**.

22

The **default port for SSH client connections is 22**; to change this default, enter a port number between 1024 and 32,767.

11. What is Docker file commands

A Dockerfile is **a text document that contains all the commands a user could call on the command line to assemble an image**. Using docker build users can create an automated build that executes several command-line instructions in succession.

12. How to create a link between two containers

Network port mappings are not the only way Docker containers can connect to one another. Docker also has a linking system that allows you to link multiple containers together and send connection information from one to another. When containers are linked, information about a source container can be sent to a recipient

1. Create an external network with docker network create <network name>
2. In each of your docker-compose. yml configure the default network to use your externally created network with the networks top-level key.
3. You can use either the service name or container name to connect between containers.

13. How to port forward in docker file

Port forwarding or port mapping **redirects a communication request from one IP address and port number combination to another**. Through port forwarding, services are exposed to the applications residing outside of the host's internal network.

How to Expose Ports in Docker

1. Add an EXPOSE instruction in the Dockerfile.
2. Use the –expose flag at runtime to expose a port.
3. Use the -p flag or -P flag in the Docker run string to publish a port.

14. Difference between small 'p' and capital 'P'(Docker)

```
-P       : Publish all exposed ports to the host interfaces
-p=[]     : Publish a container's port or a range of ports to the host
           format: ip:hostPort:containerPort | ip::containerPort | hostPort:containerPort | containerPort
           Both hostPort and containerPort can be specified as a
           range of ports. When specifying ranges for both, the
           number of container ports in the range must match the
           number of host ports in the range, for example:
              -p 1234-1236:1234-1236/tcp
```

15. How to create a job in Jenkins

1. Step 1) Login to Jenkins. ...
2. Step 2) Create New Item. ...
3. Step 3) Enter Item details. ...
4. Step 4) Enter Project details. ...
5. Step 5) Enter repository URL. ...
6. Step 6) Tweak the settings. ...
7. Step 7) Save the project. ...
8. Step 8) Build Source code.
9. Check the status
10. see the console output

Summary

11. Jenkins Freestyle Project is a repeatable build job, script, or pipeline that contains steps and post-build actions. It is an improved job or task that can span multiple operations.
12. The types of actions you can perform in a build step or post-build action are quite limited. There are many standard plugins available within a Jenkins Freestyle Project to help you overcome this problem.
13. Freestyle build Jenkins jobs are highly flexible and easy-to-use. You can use it for any type of project; it is easy to set up, and many of its options appear in other build Jenkins jobs.
14. If your GitHub repository is private, Jenkins will first validate your login credentials with GitHub and only then pull the source code from your GitHub repository.

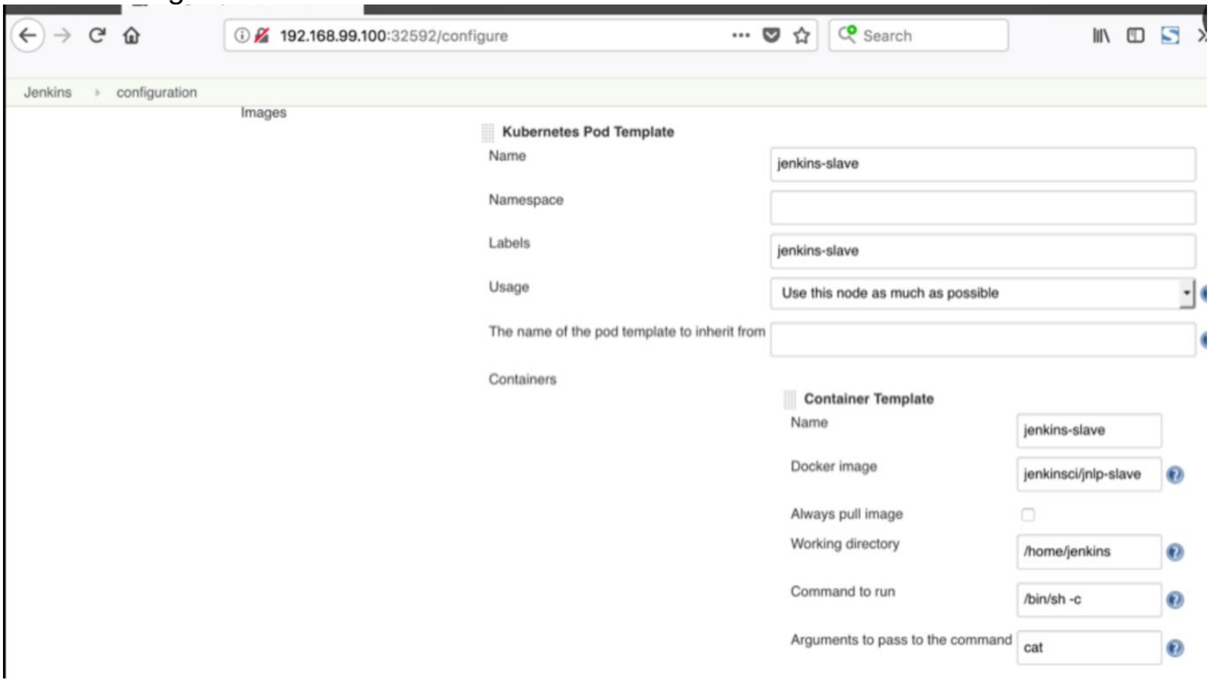
16. What plugins are u used in your project

Top 25 Jenkins plugins for productive DevOps

Setup & Scaling

1. Kubernetes

The “Kubernetes” plugin is great for automating build agents on a Kubernetes cluster. Essentially, the plugin will dynamically create Kubernetes Pods that house a build agent that has started and will stop the agent once the build has completed.
Navigating to **Manage Jenkins -> Configure System -> Cloud -> Kubernetes** will allow you to easily configure this free plugin. Note that if Jenkins is running on the cluster, the default configuration values can be used instead.



Kubernetes Pod Template

Name: jenkins-slave

Namespace:

Labels: jenkins-slave

Usage: Use this node as much as possible

The name of the pod template to inherit from:

Containers

Container Template

Name: jenkins-slave

Docker image: jenkinsci/jnlp-slave

Always pull image: □

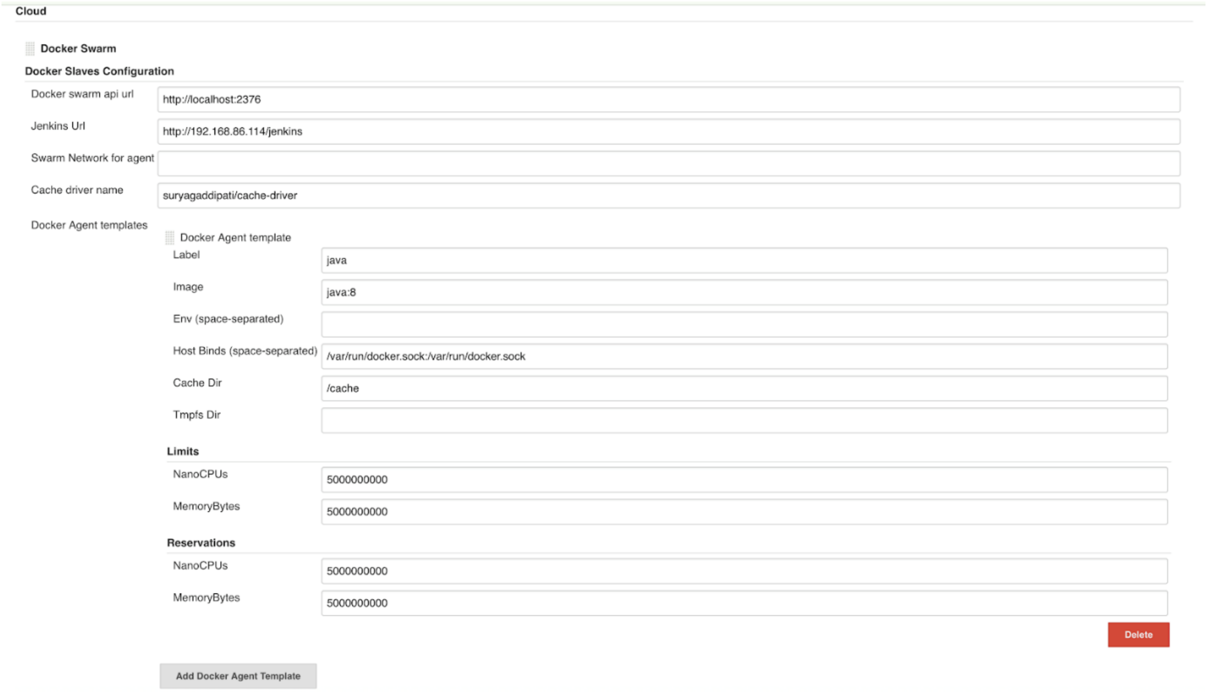
Working directory: /home/jenkins

Command to run: /bin/sh -c

Arguments to pass to the command: cat

2. Swarm

This interesting plugin is useful if you plan on using Docker Swarm. **It helps make life easier by allowing you to add worker nodes to a Jenkins master node effectively creating a cluster** and making scalability much easier.
“Swarm” also requires a client CLI application to be installed in order to have the secondary nodes join the primary node. Both the plugin and the CLI application are open source software.



Cloud

Docker Swarm

Docker Slaves Configuration

Docker swarm api url: http://localhost:2376

Jenkins Url: http://192.168.86.114/jenkins

Swarm Network for agent:

Cache driver name: suryagaddipati/cache-driver

Docker Agent templates

Docker Agent template

Label: java

Image: java:8

Env (space-separated):

Host Binds (space-separated): /var/run/docker.sock:/var/run/docker.sock

Cache Dir: /cache

Tmpfs Dir:

Limits

NanoCPUs: 5000000000

MemoryBytes: 5000000000

Reservations

NanoCPUs: 5000000000

MemoryBytes: 5000000000

3. Amazon Elastic Container Service

A plugin that deploys build agents to an existing Amazon ECS cluster. These builds run within separate Docker containers that are removed upon completion of the build. This plugin is free to download on your Jenkins instance, however, an Amazon AWS account is required. As of the time of this writing, the plugin is looking for new maintainers.

4. Azure Container Service

Similar to Amazon ECS, this plugin requires an existing cluster on Azure. Keep in mind that **Azure Container Services is being deprecated by Microsoft, but this plugin still supports it as well as Azure Kubernetes Service**.

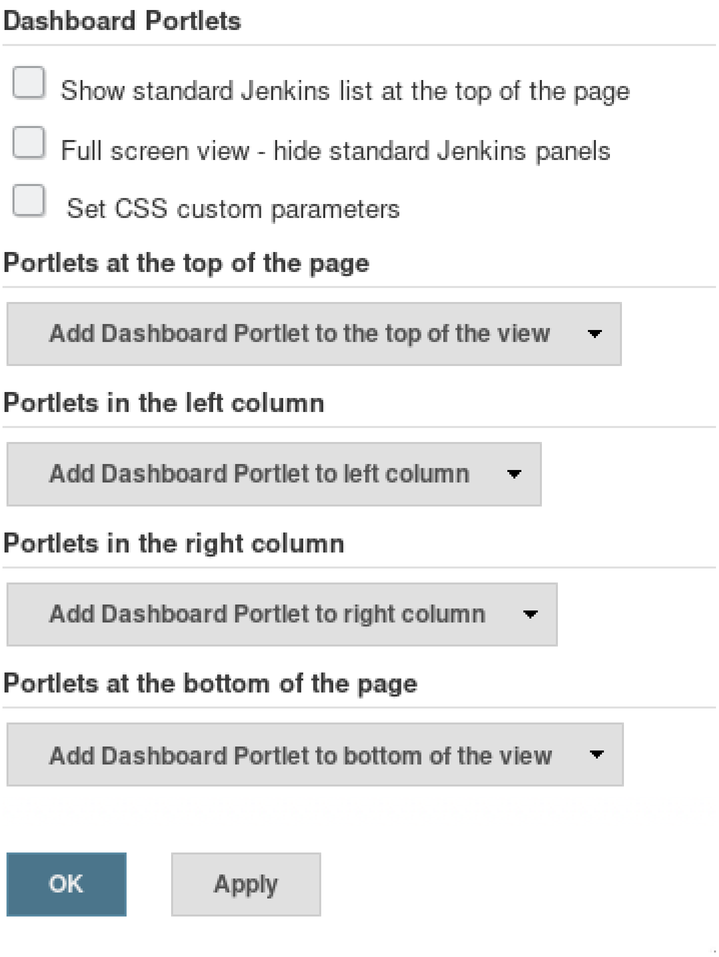
Like Amazon ECS, this plugin is free to use, but it does require an Azure account.

Productivity

5. Dashboard View

Dashboard View enables you to create a customized view within the Jenkins dashboard. The user is able to select which jobs they want to include in the view as well as the different portlets.

Creating a new view in this open source plugin is very easy and only requires a few button clicks.



Dashboard Portlets

□ Show standard Jenkins list at the top of the page

□ Full screen view - hide standard Jenkins panels

□ Set CSS custom parameters

Portlets at the top of the page

Add Dashboard Portlet to the top of the view

Portlets in the left column

Add Dashboard Portlet to left column

Portlets in the right column

Add Dashboard Portlet to right column

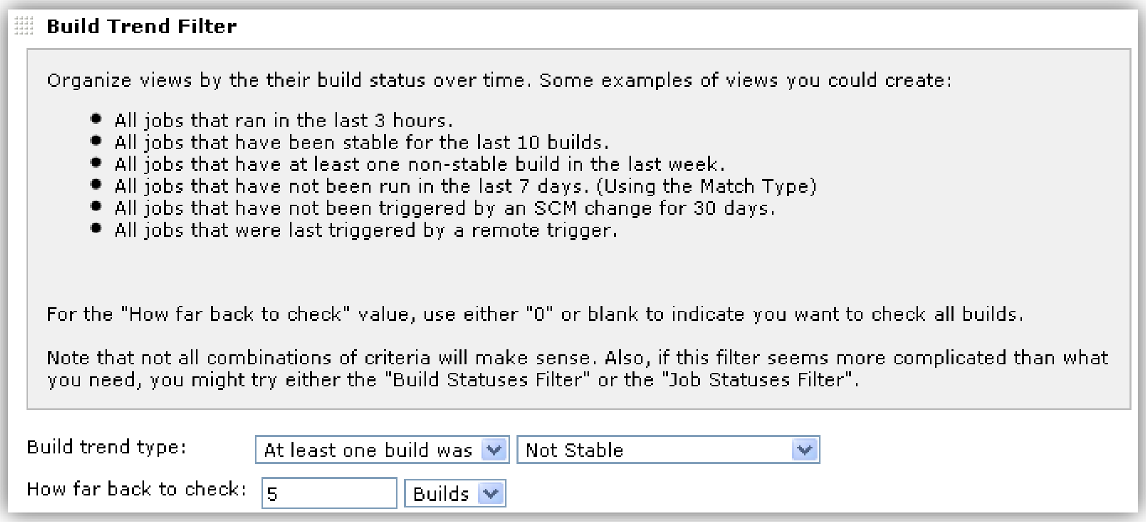
Portlets at the bottom of the page

Add Dashboard Portlet to bottom of the view
OK
Apply

6. View Job Filters

The "View Job Filters" plugin lets you choose from a wide range of filters to help manage lots of jobs. Basically, this lets you see only the jobs you want to see within a view.

An interesting feature of this free plugin is that it does have a regular expression filter. This will be useful for companies that have hundreds of jobs running.



Build Trend Filter

Organize views by the their build status over time. Some examples of views you could create:

- All jobs that ran in the last 3 hours.
- All jobs that have been stable for the last 10 builds.
- All jobs that have at least one non-stable build in the last week.
- All jobs that have not been run in the last 7 days. (Using the Match Type)
- All jobs that have not been triggered by an SCM change for 30 days.
- All jobs that were last triggered by a remote trigger.

For the "How far back to check" value, use either "0" or blank to indicate you want to check all builds.

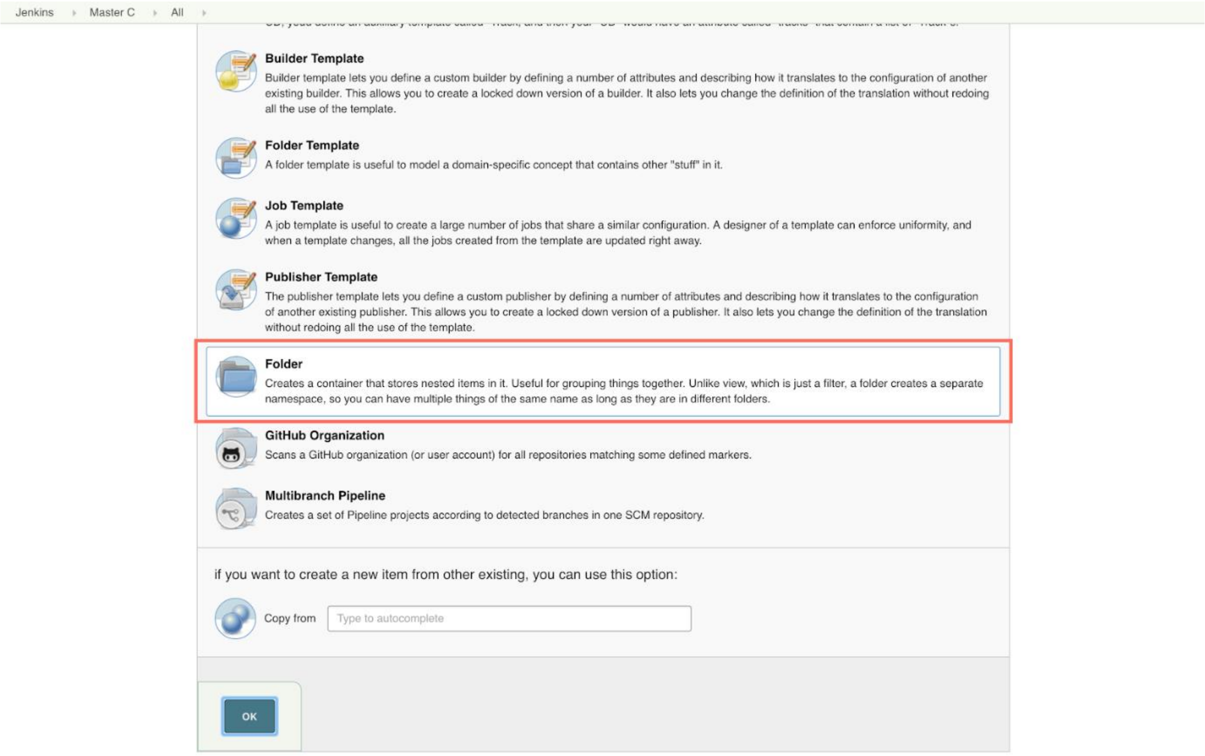
Note that not all combinations of criteria will make sense. Also, if this filter seems more complicated than what you need, you might try either the "Build Statuses Filter" or the "Job Statuses Filter".

Build trend type: At least one build was | Not Stable

How far back to check: 5 | Builds

7. Folders

Tired of looking at a giant list of jobs? The "Folders" plugin allows you to organize them into your own customized folder structure.



CD, you'd define an auxiliary template called "Track", and then your "CD" would have an attribute called "tracks" that contain a list of "Track"s.

Builder Template
Builder template lets you define a custom builder by defining a number of attributes and describing how it translates to the configuration of another existing builder. This allows you to create a locked down version of a builder. It also lets you change the definition of the translation without redoing all the use of the template.

Folder Template
A folder template is useful to model a domain-specific concept that contains other "stuff" in it.

Job Template
A job template is useful to create a large number of jobs that share a similar configuration. A designer of a template can enforce uniformity, and when a template changes, all the jobs created from the template are updated right away.

Publisher Template
The publisher template lets you define a custom publisher by defining a number of attributes and describing how it translates to the configuration of another existing publisher. This allows you to create a locked down version of a publisher. It also lets you change the definition of the translation without redoing all the use of the template.

Folder
Creates a container that stores nested items in it. Useful for grouping things together. Unlike view, which is just a filter, a folder creates a separate namespace, so you can have multiple things of the same name as long as they are in different folders.

GitHub Organization
Scans a GitHub organization (or user account) for all repositories matching some defined markers.

Multibranch Pipeline
Creates a set of Pipeline projects according to detected branches in one SCM repository.

if you want to create a new item from other existing, you can use this option:

Copy from Type to autocomplete

8. Jira

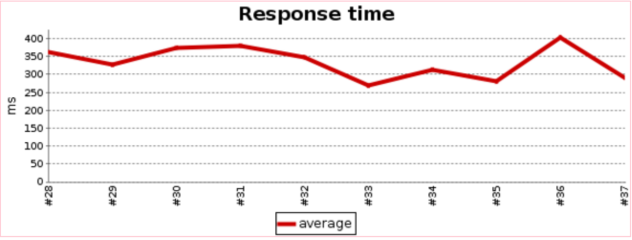
"Jira" is an open source plugin that does exactly what it says. **Once installed, you can integrate your Jenkins instance with Atlassian Jira Software**.
It is recommended, when using this plugin, to use a Jira service account instead of a personal account.

Performance

9. Performance

With this free plugin, you can run performance reports for your favorite test suites. Supported suites include JUnit, JMeter, Taurus, and others.
Setting up a performance test is a very easy process that requires you to add a build step to run a performance test.

Performance Breakdown by URI: stats.xml



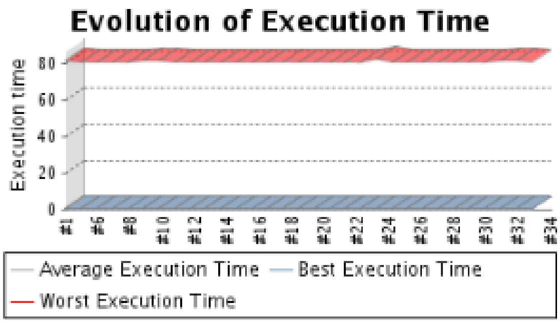
Response time trends for build: "perf-plugin-test #37"

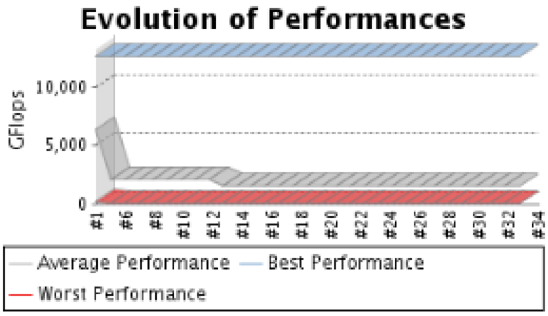
URI	Samples	Average (ms)	Median (ms)	Line90 (ms)	Minimum (ms)	Maximum (ms)	Http Code	Previous Http Code	Errors (%)	Average (KB)	Total (KB)
__localhost_8080_	208 +28	540 -208	486 -255	844	202	2523			0.0 % -1.111 %		
__localhost_8080_api_	208 +33	42 -5	37 -8	72	14	258			0.0 % 0.0 %		
All URIs	416 +61	291 -111	216 -119	763	14	2523			0.0 % -0.563 %	479903.22	1.99639739E8

10. Performance Publisher

The "Performance Publisher" plugin generates global and trend reports that can be used for test result analysis. The really cool part is that it works with any test suite.

A new version of this plugin hasn’t been released for a couple of years and it is currently up for adoption. However, based on the amount of downloads, there are still plenty of users.

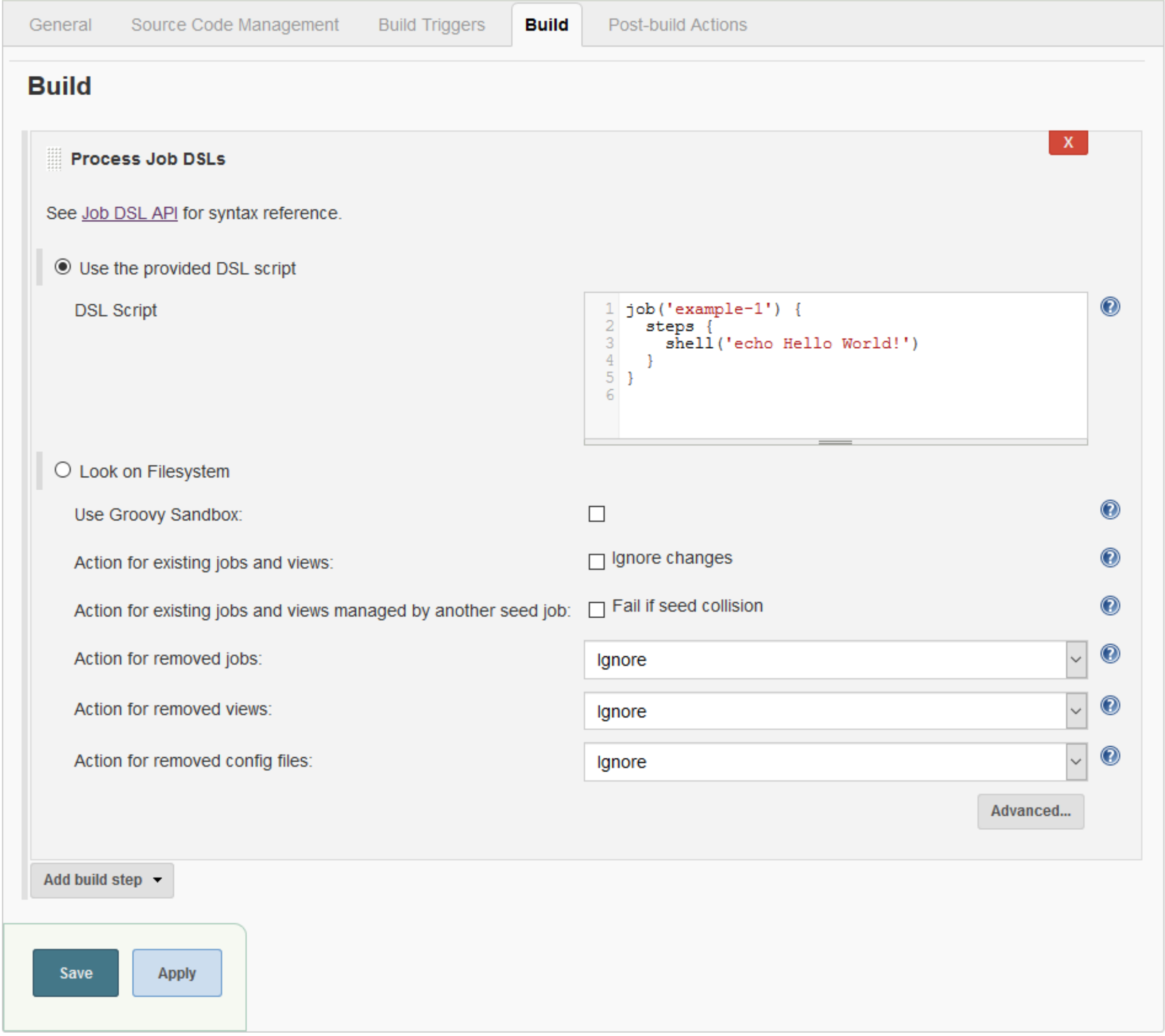




Pipeline / Flow

11. Job DSL

The idea behind this plugin is that when you have a large number of jobs to manage, using the User Interface can be a tedious task. Therefore, **this plugin helps you easily define jobs using a Groovy Based Language** (a scripting language).
One thing to keep in mind, is that the original plugin was deprecated. However, the newer version receives a ton of support on Github.



Build

Process Job DSLs

See Job DSL API for syntax reference.

● Use the provided DSL script

DSL Script

```
job('example-1') {
  steps {
    shell('echo Hello World!')
  }
}
```

○ Look on Filesystem

Use Groovy Sandbox: □

Action for existing jobs and views: □ Ignore changes

Action for existing jobs and views managed by another seed job: □ Fail if seed collision

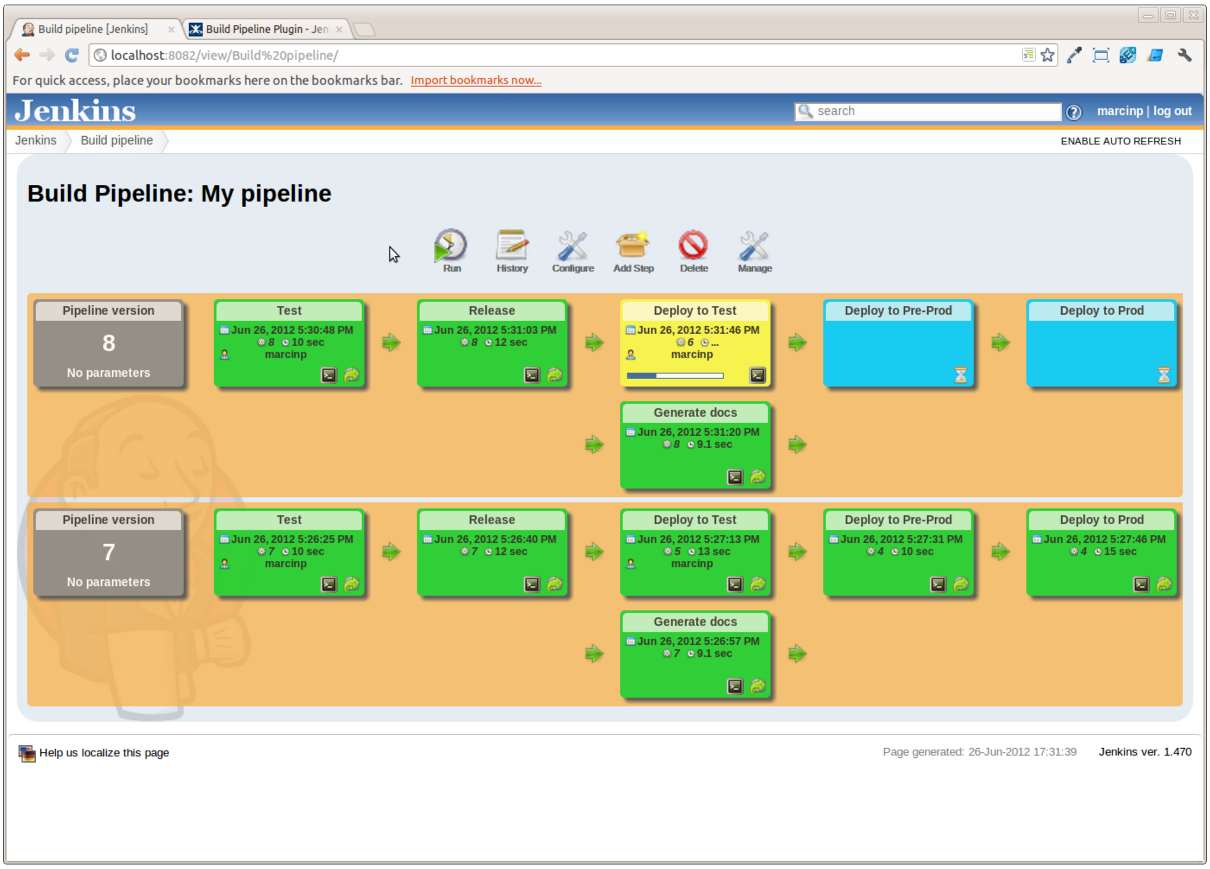
Action for removed jobs: Ignore

Action for removed views: Ignore

Action for removed config files: Ignore

12. Build Pipeline

“Build Pipeline” is another interesting plugin because it gives you a view of all the jobs within your build pipeline. It also shows all the connected jobs that are upstream and downstream. What’s all really cool is if you have any jobs that require intervention before they run, manual triggers can be defined.
At the time of this writing, the plugin has not been updated in a number of years and the current version contains a Stored XSS vulnerability.


Build pipeline [Jenkins]
Build Pipeline Plugin - Jen
localhost:8082/view/Build%20pipeline/
For quick access, place your bookmarks here on the bookmarks bar. Import bookmarks now...
Jenkins
search
marcinp | log out
Jenkins
Build pipeline
ENABLE AUTO REFRESH
Run
History
Configure
Add Step
Delete
Manage
Help us localize this page

Build Pipeline: My pipeline

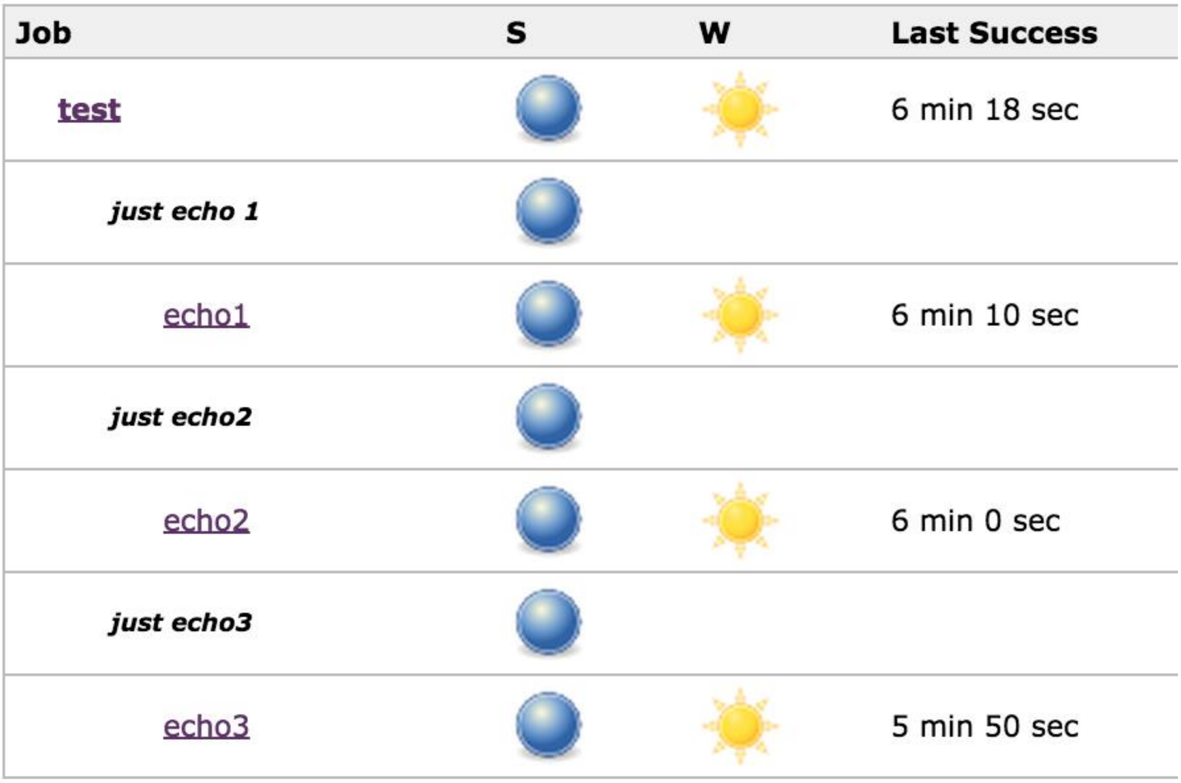
Pipeline version 8 — No parameters
Test — Jun 26, 2012 5:30:48 PM — 8 — 10 sec — marcinp
Release — Jun 26, 2012 5:31:03 PM — 8 — 12 sec
Deploy to Test — Jun 26, 2012 5:31:46 PM — 6 — ... — marcinp
Deploy to Pre-Prod
Deploy to Prod
Generate docs — Jun 26, 2012 5:31:20 PM — 8 — 9.1 sec

Pipeline version 7 — No parameters
Test — Jun 26, 2012 5:26:25 PM — 7 — 10 sec — marcinp
Release — Jun 26, 2012 5:26:40 PM — 7 — 12 sec
Deploy to Test — Jun 26, 2012 5:27:13 PM — 5 — 13 sec — marcinp
Deploy to Pre-Prod — Jun 26, 2012 5:27:31 PM — 4 — 10 sec
Deploy to Prod — Jun 26, 2012 5:27:46 PM — 4 — 15 sec
Generate docs — Jun 26, 2012 5:26:57 PM — 7 — 9.1 sec

Page generated: 26-Jun-2012 17:31:39 Jenkins ver. 1.470

13. Multijob

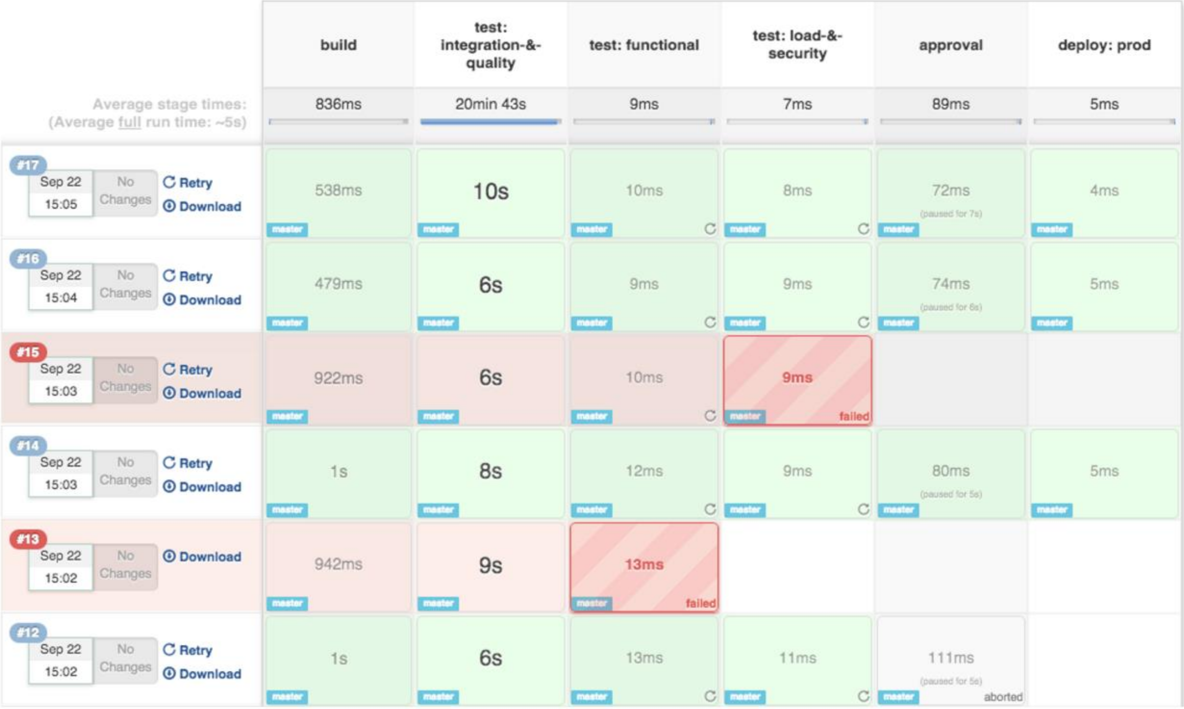
“Multijob” is a free plugin that is useful for cleaning up messes with chain definitions from upstream and downstream jobs. It’s also handy if you want to create a hierarchy of jobs that need to be executed either sequentially or in parallel.



Job	S	W	Last Success
test	●	☀	6 min 18 sec
just echo 1	●		
echo1	●	☀	6 min 10 sec
just echo2	●		
echo2	●	☀	6 min 0 sec
just echo3	●		
echo3	●	☀	5 min 50 sec

14. Pipeline

“Pipeline” is actually a group of plugins that are used for creating continuous integration pipelines. The interesting part is that these pipelines are written by the user in a domain-specific language.
Like most Jenkins plugins, “Pipeline” is open source and was originally known as “Workflow”.



	build	test: integration-&-quality	test: functional	test: load-&-security	approval	deploy: prod
Average stage times: (Average full run time: ~5s)	836ms	20min 43s	9ms	7ms	89ms	5ms
#17 Sep 22 15:05 No Changes Retry Download	538ms master	10s master	10ms master	8ms master	72ms (paused for 7s) master	4ms master
#16 Sep 22 15:04 No Changes Retry Download	479ms master	6s master	9ms master	9ms master	74ms (paused for 6s) master	5ms master
#15 Sep 22 15:03 No Changes Retry Download	922ms master	6s master	10ms master	9ms master failed		
#14 Sep 22 15:03 No Changes Retry Download	1s master	8s master	12ms master	9ms master	80ms (paused for 5s) master	5ms master
#13 Sep 22 15:02 No Changes Download	942ms master	9s master	13ms master failed			
#12 Sep 22 15:02 No Changes Retry Download	1s master	6s master	13ms master	11ms master	111ms (paused for 5s) master aborted	

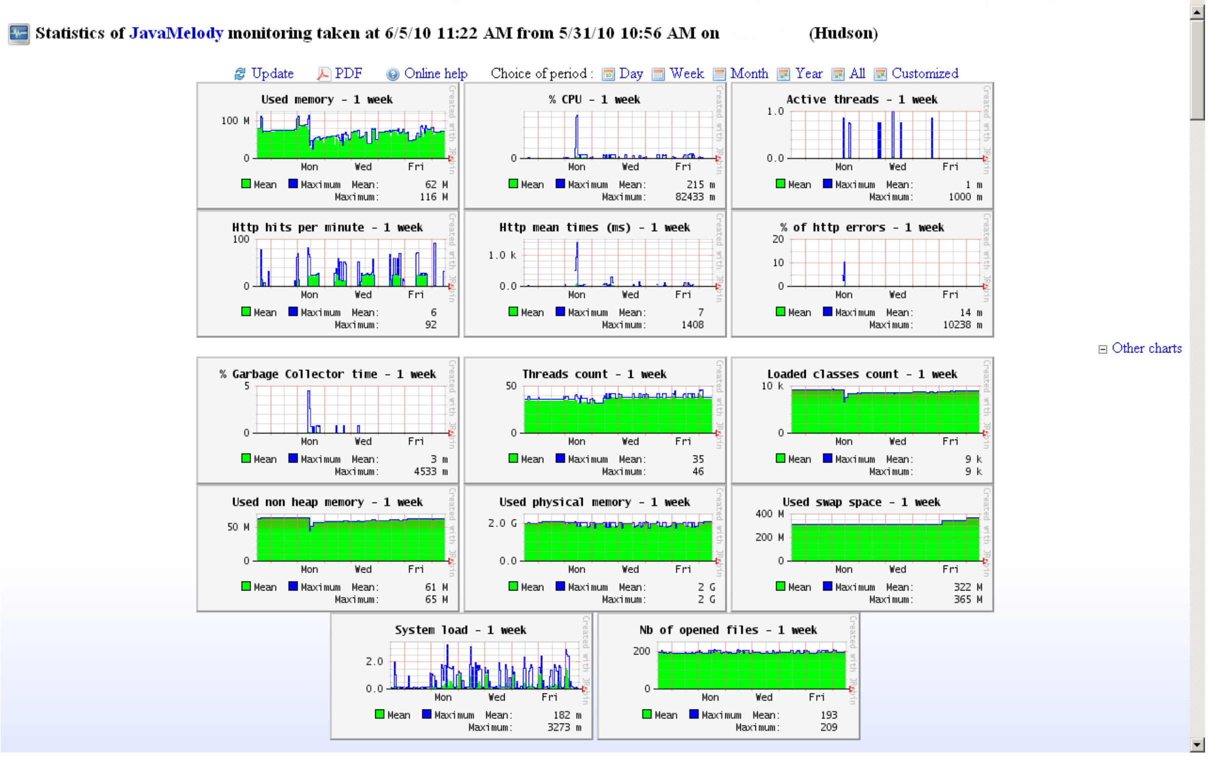
Monitoring & Alerting

15. Monitoring

Since Jenkins was written in Java it only makes sense that the “Monitoring” plugin uses JavaMelody. In a nutshell, **JavaMelody is an open source monitoring tool for Java and Java EE applications**. This plugin monitors errors, issues, security, HTTP sessions, etc.
To access the monitoring reports, navigate to the url

```
http://host/monitoring
```

after installation.



Statistics of JavaMelody monitoring taken at 6/5/10 11:22 AM from 5/31/10 10:56 AM on (Hudson)

Update
PDF
Online help
Choice of period :
Day
Week
Month
Year
All
Customized
Used memory - 1 week
Mean: 62 M Maximum: 116 M
% CPU - 1 week
Mean: 215 m Maximum: 82433 m
Active threads - 1 week
Mean: 1 m Maximum: 1000 m
Http hits per minute - 1 week
Mean: 6 Maximum: 92
Http mean times (ms) - 1 week
Mean: 7 Maximum: 1408
% of http errors - 1 week
Mean: 14 m Maximum: 10238 m
Other charts
% Garbage Collector time - 1 week
Mean: 3 m Maximum: 4533 m
Threads count - 1 week
Mean: 35 Maximum: 46
Loaded classes count - 1 week
Mean: 9 k Maximum: 9 k
Used non heap memory - 1 week
Mean: 61 M Maximum: 65 M
Used physical memory - 1 week
Mean: 2 G Maximum: 2 G
Used swap space - 1 week
Mean: 322 M Maximum: 365 M
System load - 1 week
Mean: 182 m Maximum: 3273 m
Nb of opened files - 1 week
Mean: 193 Maximum: 209
Mon
Wed
Fri
🟩 Mean
🟦 Maximum

16. Disk-usage

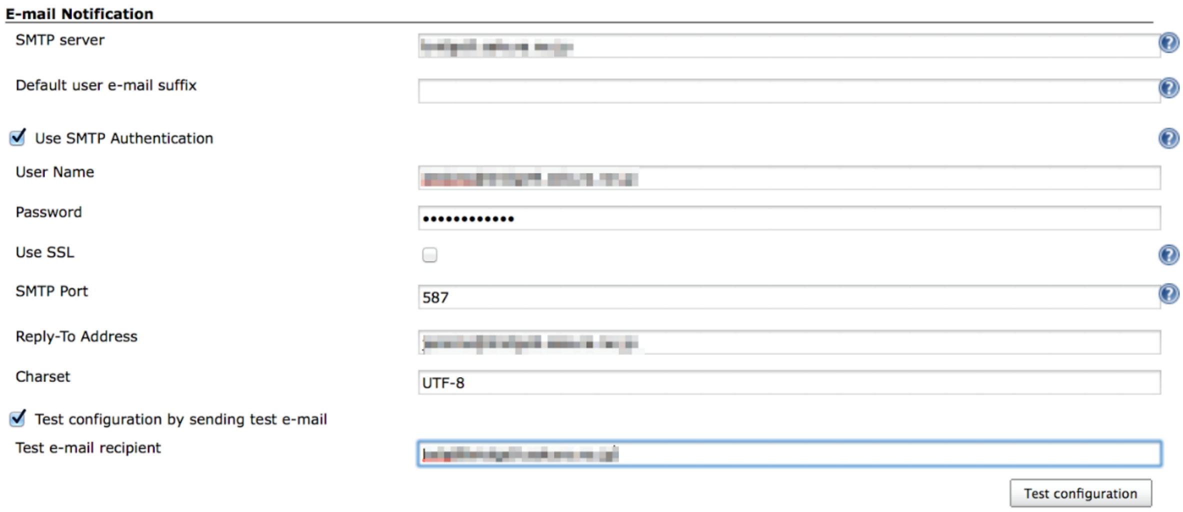
It is very important for you to know how much storage you have left in your Jenkins instance. **This plugin shows you how much disk space is being used by your projects**.
Although the plugin hasn’t been updated in a few years, it is still a very handy tool to have in your toolbox. New maintainers are being requested for this plugin as well.

17. Metrics

The “Metrics” plugin uses the Dropwizard Metrics API to conduct standard health checks and gather standard metrics on plugins installed on your Jenkins instance. Both the “Metrics” plugin and Dropwizard Metrics API are open source tools that receive active support on Github.

18. Mailer

Once configured and added to the Post-Build action of the Jenkins job, the “Mailer” plugin will send you emails based on the job results.
If you do not have an SMTP server, Jenkins will use the default server that is built into Jenkins.



E-mail Notification

SMTP server: [redacted]

Default user e-mail suffix:

☑ Use SMTP Authentication

User Name: [redacted]

Password: ••••••••••••

Use SSL: □

SMTP Port: 587

Reply-To Address: [redacted]

Charset: UTF-8

☑ Test configuration by sending test e-mail

Test e-mail recipient: [redacted]

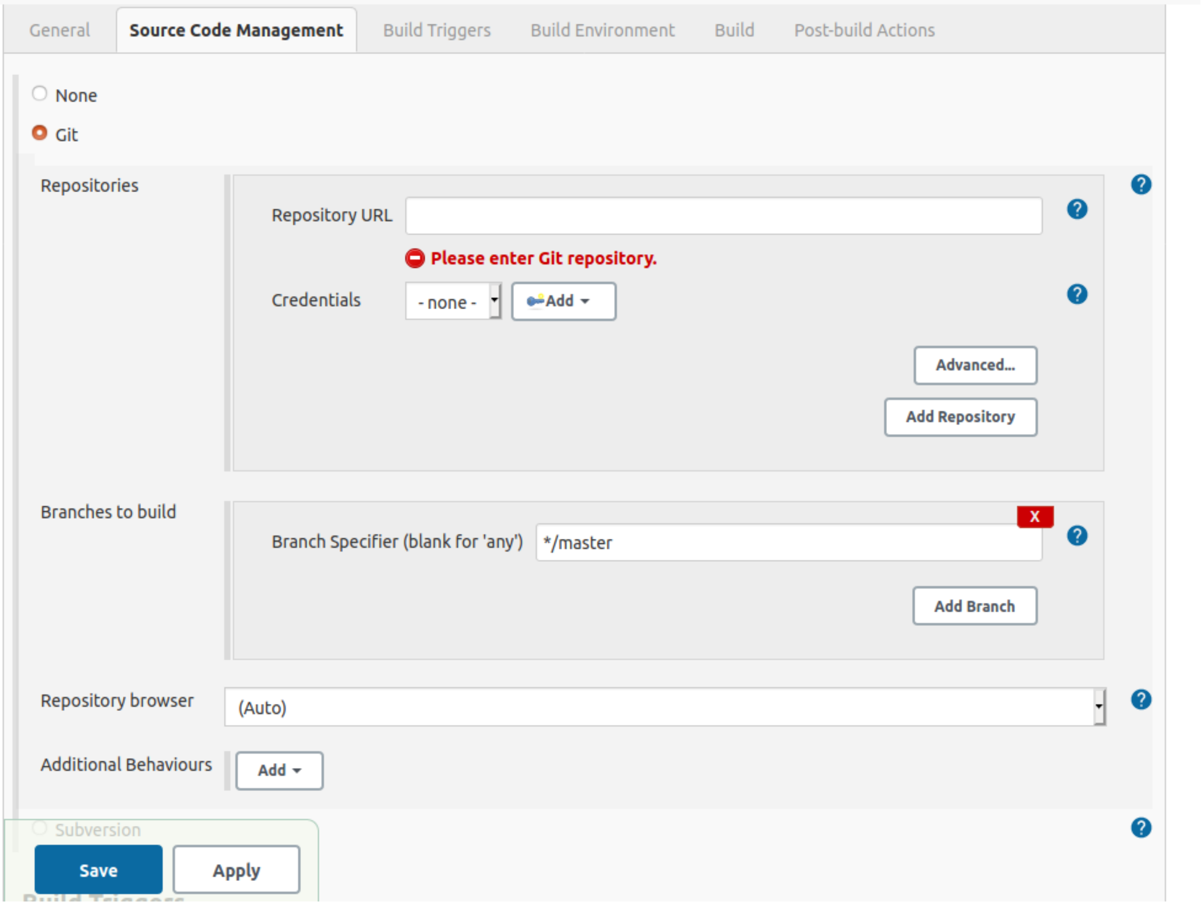
Source Control Management (SCM)

19. SCM API

“SCM API” is a plugin that integrates with source control management systems. The built-in extension points allow developers to receive event notifications from SCM systems and easily browse through repositories and organizations.

20. Git

The “Git” plugin allows jobs to connect to remote repositories and run git operations against them.
From a security standpoint, the plugin utilizes the “Jenkins credentials plugin”. Therefore, the credential types secret text, secret file and certificates are not supported.



○ None

● Git

Repositories

Repository URL:

Please enter Git repository.

Credentials: - none -

Branches to build

Branch Specifier (blank for 'any'): */master

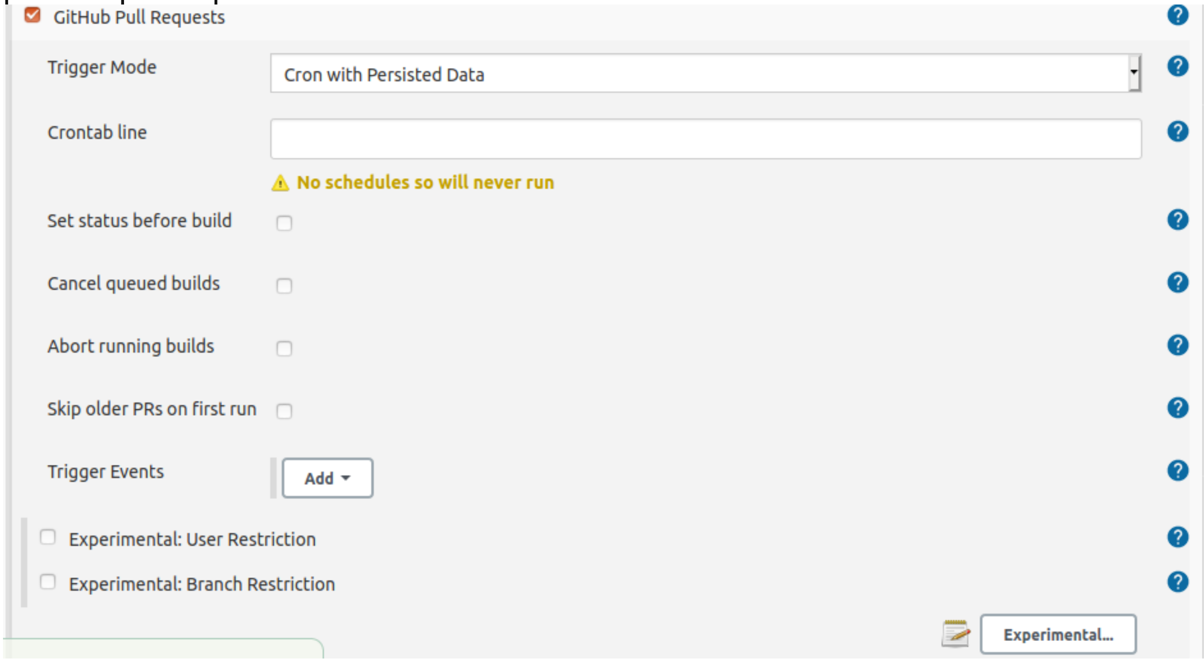
Repository browser: (Auto)

Additional Behaviours

○ Subversion

21. GitHub Integration

With this plugin, you can integrate your Jenkins instance directly with Github. That means you can pull down code and files from Github to Jenkins, scheduled builds and process pull requests.



☑ GitHub Pull Requests

Trigger Mode: Cron with Persisted Data

Crontab line:

⚠ No schedules so will never run

Set status before build □

Cancel queued builds □

Abort running builds □

Skip older PRs on first run □

Trigger Events: Add ▾

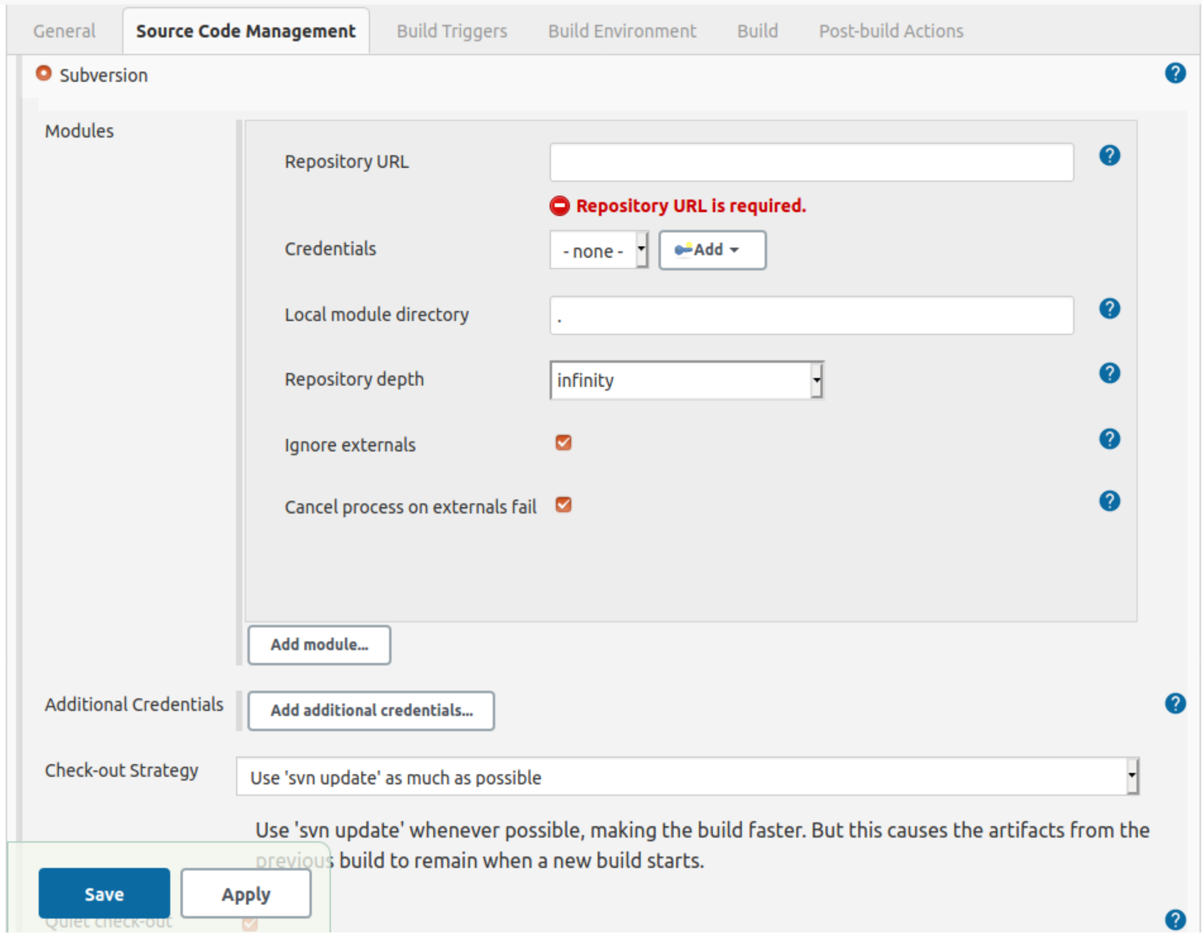
□ Experimental: User Restriction

□ Experimental: Branch Restriction

22. Subversion

This plugin adds Subversion repositories as an option in the SCM section of the job configurations screen.

It also allows the use of server certificates to connect to your repository.



● Subversion

Modules

Repository URL:

Repository URL is required.

Credentials: - none - Add ▾

Local module directory: .

Repository depth: infinity

Ignore externals ☑

Cancel process on externals fail ☑

Add module...

Additional Credentials: Add additional credentials...

Check-out Strategy: Use 'svn update' as much as possible

Use 'svn update' whenever possible, making the build faster. But this causes the artifacts from the previous build to remain when a new build starts.

Quiet check-out ☑

Tests & Analysis

23. Test Results Analyzer

The “Test Results Analyzer” is another open source plugin that takes away the pain of having to search through every individual build report. Essentially, it shows the result history of the builds in a tabular format that can be filtered based on what you want to see.

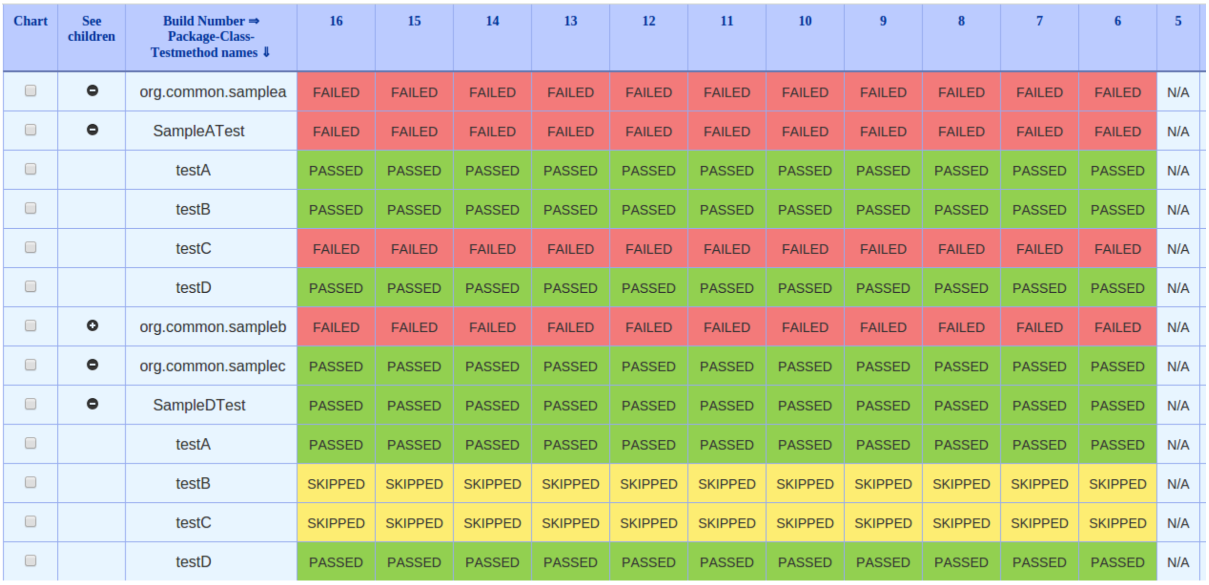


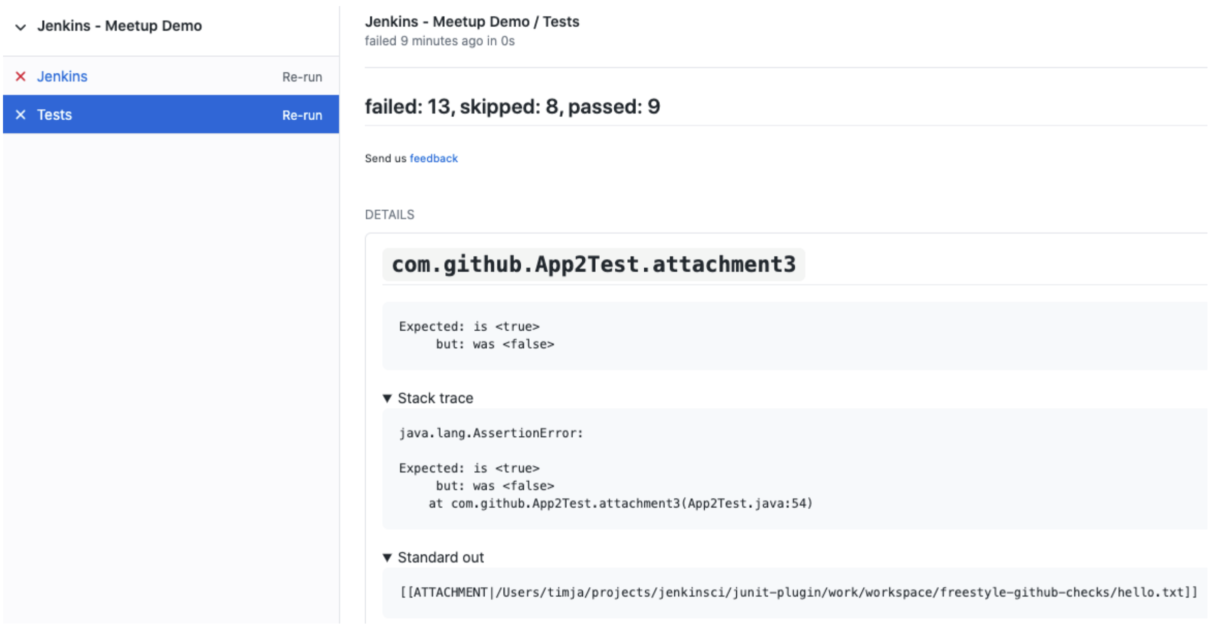
Chart	See children	Build Number ⇒ Package-Class-Testmethod names ⇓	16	15	14	13	12	11	10	9	8	7	6	5
□	⊖	org.common.samplea	FAILED	FAILED	FAILED	FAILED	FAILED	FAILED	FAILED	FAILED	FAILED	FAILED	FAILED	N/A
□	⊖	SampleATest	FAILED	FAILED	FAILED	FAILED	FAILED	FAILED	FAILED	FAILED	FAILED	FAILED	FAILED	N/A
□		testA	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	N/A
□		testB	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	N/A
□		testC	FAILED	FAILED	FAILED	FAILED	FAILED	FAILED	FAILED	FAILED	FAILED	FAILED	FAILED	N/A
□		testD	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	N/A
□	⊕	org.common.sampleb	FAILED	FAILED	FAILED	FAILED	FAILED	FAILED	FAILED	FAILED	FAILED	FAILED	FAILED	N/A
□	⊖	org.common.samplec	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	N/A
□	⊖	SampleDTest	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	N/A
□		testA	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	N/A
□		testB	SKIPPED	SKIPPED	SKIPPED	SKIPPED	SKIPPED	SKIPPED	SKIPPED	SKIPPED	SKIPPED	SKIPPED	SKIPPED	N/A
□		testC	SKIPPED	SKIPPED	SKIPPED	SKIPPED	SKIPPED	SKIPPED	SKIPPED	SKIPPED	SKIPPED	SKIPPED	SKIPPED	N/A
□		testD	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	PASSED	N/A

24. bootstrapped-multi-test-results-report

Instead of staring at boring looking test results, this plugin makes them look pretty by using the Bootstrap HTML, CSS, and JS library. Support for test suites includes Cucumber, JUnit, RSpec, and TestNG
It has been a few years since the last release, but the plugin still receives a decent amount of downloads.

25. JUnit

The “JUnit” plugin is a free tool that provides graphical visualizations for test results. It also provides a user interface for viewing test reports and failures.
By installing the “Github Checks Plugin”, “JUnit will run checks against your Github projects.



Jenkins - Meetup Demo / Tests
failed 9 minutes ago in 0s

failed: 13, skipped: 8, passed: 9

Send us feedback

DETAILS

`com.github.App2Test.attachment3`

```
Expected: is <true>
     but: was <false>
```

▼ Stack trace

```
java.lang.AssertionError:

Expected: is <true>
     but: was <false>
    at com.github.App2Test.attachment3(App2Test.java:54)
```

▼ Standard out

```
[[ATTACHMENT|/Users/timja/projects/jenkinsci/junit-plugin/work/workspace/freestyle-github-checks/hello.txt]]
```

Out of the box, Jenkins is an amazing tool. But when you add a few of these plugins, it becomes even better. Jenkins plugins give you the freedom to connect to SCMs, analyze test results, monitor jobs, build continuous deployment pipelines, etc.

The ones we have outlined here are considered essential. This is because they have the capabilities to take your DevOps environment to the next level and make it far more productive. Think we missed one? Tell us in the comments!

17. How to create a role in ansible

To create ansible role, **use ansible-galaxy init <role_name> to create the role directory structure**.

Ie **Writing an Ansible Role**
Writing an Ansible Role is fairly easy. Follow the aforementioned steps to write a Role:

1. Initialise the role structure using the command:

```
ansible-galaxy init &lt;role-name&gt;
```

2. Go to role directory using:

```
cd &lt;role-name&gt;
```

3. Initialise for Git:

```
git init
```

4. Do the necessary changes required to add role functionality.

5. Add files to Git using the command:

```
git add *
```

6. Commit the changes to Git:

```
git commit -m "Initial commit for &lt;role-name&gt;"
```

7. Set your public repository origin:

```
git remote add origin &lt;github-url&gt;
```

8. Push the changes to the repository using the command:

```
git push origin master
```

9. Release a version in the releases tab of GitHub which will be published in Galaxy as a Role.

Publishing an Ansible Role

Go to https://galaxy.ansible.com
Log in to Galaxy using GitHub credentials.

18. what are challenges you are faced

1. Environment provisioning
2. Manual testing
3. No DevOps center of excellence
4. Test data
5. Manual deployments
6. Planning in a DevOps environment
7. DevOps and suppliers
8. DevOps and governance
9. No integrated tools architecture
10. Manual releases
11. No DevOps metrics
12. DevOps and team composition
13. DevOps and regulatory compliance
14. No service virtualization
15. DevOps and specialist skills
16. Traceability across the DevOps landscape
17. Large releases
18. Inconsistent environments
19. Agile confined to developers
20. Limited transparency
21. Manual processes

22. Collaboration between development and operations

23. No DevOps vision or strategy

24. No production-like environments

25. Waste in existing processes

26. Limited customer feedback

27. Elicitation of nonfunctional requirements

28. Collaboration across all IT disciplines

29. Collaboration between business and IT

30. No standard SCM repository

19. Are u faced a performance issue

Application performance monitoring has traditionally focused on monitoring and analyzing just applications and the infrastructure that hosts them.

In today's DevOps-centric world, however, where new application releases and updates are delivered continuously using CI/CD pipelines, monitoring CI/CD operations has become a third key pillar for optimizing overall application performance. Even the best-written code or the most flawless application will result in a poor user experience if problems in the CI/CD pipeline prevent smooth and continuous deployment.

Likewise, if CI/CD problems make it difficult to assess the performance impact of code or configuration changes, you'll be shooting in the dark and struggling to optimize performance.

Here's a primer on how to monitor the CI/CD delivery pipeline and how to correlate that data with other metrics in order to achieve optimal overall performance of your applications.

Why CI/CD Monitoring Matters

The CI/CD pipeline is distinct from the software environment that hosts your application, but it's nonetheless linked inextricably to it. A healthy pipeline is one that allows your team to write, build, test, and deploy code and configuration changes into the production environment on a continuous basis.

An unhealthy CI/CD pipeline can hamper your ability to achieve optimal application performance in a variety of ways. For example:

Slow deployments:

If your CI/CD operations are slow and you are unable to push out new releases quickly, you may not be able to deploy fixes to performance bugs before they become critical problems for your end-users.

Testing completeness:

Inefficient CI/CD operations (such as slow builds, or messy handoffs of new code from developers to the software testing team) hamper your inability to test software completely before you deploy. They force you to choose between deploying releases that haven’t been fully tested or delaying deployments while you wait on tests to complete. Neither outcome is good for end-users.

Testing coverage:

CI/CD operations issues may also make it difficult to test each release against a wide variety of configuration variables. If you don’t have as much time to test as you would ideally, you may have to test only for some use cases or some environment configurations, which makes it more difficult to ensure adequate application performance for all users once code reaches production.

Technical debt:

Lack of visibility into the CI/CD process can lead to technical debt. When you can’t systematically measure the performance of each part of your CI/CD pipeline, it’s much harder to determine processes that are causing technical debt.

Deployment agility:

Total visibility into the CI/CD pipeline makes it easier to achieve deployment agility, such as the ability to deploy to a new kind of production environment (a different cloud, for instance) or to make major configuration changes to the environment. When you know exactly how each CI/CD process is going and what a successful CI/CD operation looks like, you can modify your operations with confidence, knowing that you’ll be able to assess rapidly whether the changes positively or negatively impact application health.

Adding CI/CD Monitoring to Application Performance Monitoring

To reduce the risk of problems or inefficiencies like those described above, teams should monitor CI/CD operations as closely as they monitor their applications and environment. CI/CD monitoring means collecting and analyzing metrics like the following:

Deployment frequency:

How many deploys do you successfully push out each day or week?

Deployment time:

How long does it take to execute each deployment? In other words, how long does it take to move a validated release from dev/test into production?

Lead time for changes:

When your team decides to implement a code or configuration change in the application, how long does it take to implement and deploy that change?

Mean time to recover:

When a problem that is detected in production necessitates a new release that includes a fix, how quickly is your team able to push out the fix?

Change failure rate:

How many attempted changes result in failures because the release in which they were implemented failed tests or otherwise was not deployed successfully?

Work-in-progress:

How many in-progress code or configuration changes are in your pipeline at a given time?

To deliver the greatest level of visibility, these metrics should be correlated with other data, including log analytics and traces from your application environment. For example, if tracing shows a performance problem in production that requires a code change to fix, CI/CD pipeline metrics about work-in-progress and deployment time will help predict how long it will take to implement the fix. Likewise, if you compare deployment frequency to baseline application performance metrics and notice that application performance is decreasing over time, it may be a sign that you are deploying so frequently that you're cutting corners on quality.

Conclusion

Gaining complete visibility into application performance requires monitoring not just application environments themselves, but also the CI/CD pipelines that power them. By correlating CI/CD data with other metrics, traces, and log analytics, you put yourself in the strongest position to optimize application performance and delight your users, even in fast-moving continuous delivery chains. Start by taking a real-time, NoSample™ full-fidelity approach in application performance monitoring that allows for unlimited cardinality exploration with Splunk APM. Learn how faster troubleshooting, easier root cause analysis and more efficient remediation leads to happier SRE and IT teams!

20. How to launch an instance from AWS

Steps to launch an instance:

1. Initiate instance launch.
2. Step 1: Choose an Amazon Machine Image (AMI)
3. Step 2: Choose an Instance Type.
4. Step 3: Configure Instance Details.
5. Step 4: Add Storage.
6. Step 5: Add Tags.
7. Step 6: Configure Security Group.
8. Step 7: Review Instance Launch and Select Key Pair.

21. how to merge the code

To merge branches locally, use git checkout to switch to the branch you want to merge into. This branch is typically the main branch. Next, use git merge and specify the name of the other branch to bring into this branch. This example merges the jeff/feature1 branch into the main branch

22. How to write custom scripts in Maven

23. how to find a specific line in a file

24. Are u involved in ShellScripts

25. How to configure YAML file in the vagrant file

The config. yaml file **contains all the configuration settings that are needed to deploy your cluster**. From the config. yaml file, you can customize your installation by using various parameter

Infosys interview Questions

Quests I faced in the Infosys interview:

Technical:

1. Explain your roles and responsibilities.

2. Explain the stages in ur project declarative pipeline.

3. What is DevOps?

4. What is ci/cd?

5. What is the difference between DevOps and CICD?

6. What are the git commands you use?

7. Difference in git fetch and git pull?

8. What is the default port of Jenkins?

9. How to change Jenkins port no?

10. How to connect master-slave?

11. How to checkout git code in Jenkins?

12. How to take backup of ur Jenkins?

13. Do you have any script to launch slave nodes?

14. How do you resolve build failure?

15. Difference between a soft link and a hard link?

16. How to find a particular word from a file?

17. What is a stop command?

18. What is Terraform provisioner?

19. What is Terraform module?

20. Is it possible to lock Terraform module?

21. What is AWS VPN?

22. How do you select availability zones using Terraform?

23. What is the difference between S3 and other AWS storage?

24. How do you use the S3 bucket in your project?

25. What is Terraform provider versioning?

26. How do you run the docker container?

27. What is an ansible playbook?

28. Syntax of ansible-playbook

29. How u run an ansible-playbook?

30. Have you worked with any alerting monitoring tools?

Managerial:

1. Tell me about yourself

2. What are your daily routine as DevOps

3. What is the ur team size

4. How scrum call happens

5. Do you take part in requirement gathering

6. Do you take calls with onshore and clients?

7. Reason for change?

8. What is the production issue u faced?

9. What was the exact reason?

10. How did you fix it?

11. Do you write documentation

12. What other activities u do in ur project?

13. Who decides the server specification?

14. How u provision the servers?

15. Are you open to working in shifts?

L&T interview questions on Terraform

----

1. How to do rollback in terraform?

2. Which plugin do you use in terraform?

3. Explain your terraform structure?

4. How to upgrade the terraform plugin?

5. What is the state file in terraforming?

6. How do you create the state file in the CICD pipeline?

7. I have the same configurations for multiple environments but I want to change the values for environments how do u that?

8.FOr suppose multiple people work on the same configuration file, in this case how state file works?

9. How do you access the state file in our local?

10. How to apply the locks for the state file?

1. **How to remove a resource from Terraform state?**
2. **How to create a folder in the S3 bucket using terraform?**
3. **How to use dynamic resource names in Terraform?**
4. **How are data sources used in Terraform?**
5. **How to read data from a file in Terraform?**
6. **What does Terraform refresh do?**

The terraform refresh command **reads the current settings from all managed remote objects and updates the Terraform state to match**.

7. **What is the difference between modules and workspaces in Terraform?**
   In Terraform CLI, workspaces are separate instances of state data that can be used from the same working directory. You can use workspaces to manage multiple non-overlapping groups of resources with the same configuration.

   A module is **a container for multiple resources that are used together**. Every Terraform configuration has at least one module, known as its root module, which consists of the resources defined in the . tf files in the main working directory.

8. **What is the null resource in Terraform?**
   The null_resource resource **implements the standard resource lifecycle but takes no further action**. The triggers argument allows specifying an arbitrary set of values that, when changed, will cause the resource to be replaced.

9. **What is the remote-exec provisioner in Terraform? How does local-exec provisioner work in Terraform?**
   he local-exec provisioner requires no other configuration, but most other provisioners must connect to the remote system using SSH or WinRM. You must include a connection block so that Terraform will know how to communicate with the server. Terraform includes several built-in provisioners; use the navigation sidebar to view their documentation.

The remote-exec provisioner invokes a script on a remote resource after it is created. This can be used to run a configuration management tool, bootstrap into a cluster, etc. To invoke a local process, see the local-exec provisioner instead. The remote-exec provisioner requires a connection and supports both ssh and winrm.

10. **How to copy the content of an s3 bucket to another s3 bucket using Terraform?**

```
                              terraform {
 backend "s3" {
  bucket = "terraformtests"
  key    = "terraformstate.tf"
  region = "us-east-1"
 }
}


resource "aws_s3_bucket_object" "terraformtests" {
 bucket = "terraformtests"
 key    = "test/prod/1000/keys"
 source = "deploy"
 etag   = "${md5(file("keys"))}"

}
```

Short description

To copy objects from one S3 bucket to another, follow these steps:

1. Create a new S3 bucket.

2. Install and configure the AWS Command Line Interface (AWS CLI).

3. Copy the objects between the S3 buckets.

Note: Using the aws s3 ls or aws s3 sync commands on large buckets (with 10 million objects or more) can be expensive, resulting in a timeout. If you encounter timeouts because of a large bucket, then consider using Amazon CloudWatch metrics to calculate the size and number of objects in a bucket. Also, consider using S3 Batch Operations to copy the objects.

4. Verify that the objects are copied.

5. Update existing API calls to the target bucket name.

11. Should I commit Terraform State files to the git repository?

Terraform state is used to map your real world infrastructure to the resources you've defined in your Terraform configuration.

he State file keeps track of which resource configuration maps to which real world resource. The state also stores metadata about those resources such as the dependency order for creating the resources.

Ultimately you can think of Terraform state as just a big JSON array of your resources (because that's pretty much what it is).

Terraform state can contain sensitive information which should not be stored in source control. Additionally if Terraform executes on different state files (i.e on two separate machines) it might break your Terraform setup. The solution? Setup a Terraform backend.

What is sar command in Linux?

sar: **System Activity Report**. It can be used to monitor Linux system's resources like CPU usage, Memory utilization, I/O devices consumption, Network monitoring, Disk usage, process and thread allocation, battery performance, Plug and play devices, Processor performance, file system and mor

HOW SSH WORKS

The way SSH works is by **making use of a client-server model to allow for authentication of two remote systems and encryption of the data that passes between them**. SSH operates on TCP port 22 by default (though this can be changed if needed). The host (server) listens on port 22 (or any other SSH assigned port) for incoming connections.

What is SCP?

Based on SSH, SCP (**Secure Copy Protocol**) transfers files via encrypted IP-based data tunnels. It does this by moving files between local hosts and remote hosts (or two remote hosts).

What is FTP?

FTP (**File Transfer Protocol**) is the traditional way to transfer files from clients to servers. Invented in the 70s, FTP is a simple way to move files between computers via TCP/IP — the framework that connects network devices online. Here's how FTP usually works:

1. You upload files to the FTP server.
2. You send these files via TCP/IP to the FTP host.
3. The recipient receives and downloads the files.

What is FTPS?

FTPS (**File Transfer Protocol Secure**, sometimes called **FTP/SSL**) is an extension of FTP created in the late 90s. Its purpose? To add an extra tier of security to FTP. FTPS uses an SSL/TLS layer underneath FTP, which encrypts its data channels.

What is SFTP?

SFTP (**Secure File Transfer Protocol**) also originated in the late 90s as an alternative to FTP. SFTP transfers various file formats via SSH, a client-server-based protocol. Unlike FTP, this only requires a single connection and encrypts files during the transfer process, making it harder for hackers to infiltrate sensitive information.

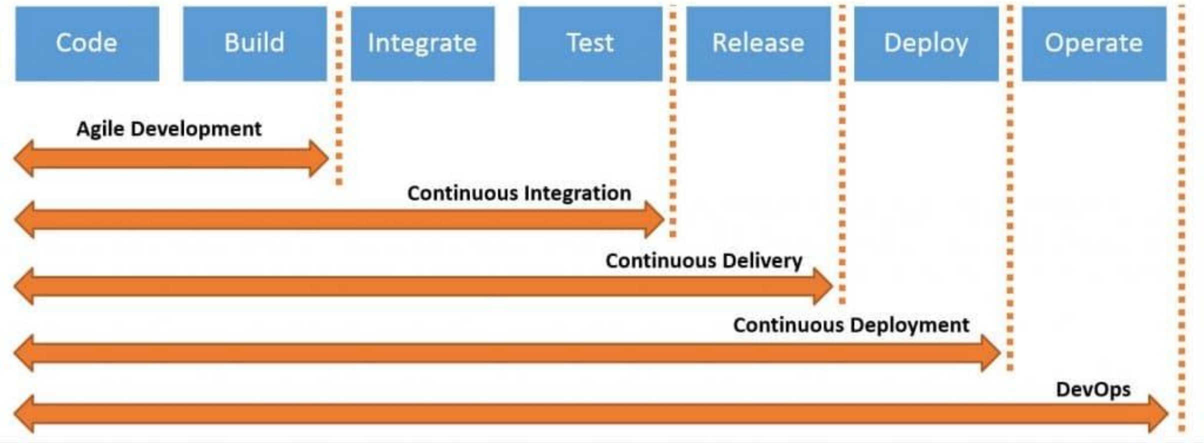
Similar to SSL, SFTP uses commands to execute the data connection when you transfer files. The recipient of your files connects to the SSH server and authenticates the server with cryptographic keys (SSH keys) or a username/password combo.

Other File Transfer Protocols

- TFTP (**Trivial File Transfer Protocol**) works on the User Datagram Transport Protocol (UDP) for file transfers. It dates back to the early 80s, and few companies use it anymore.
- MFT (**Managed File Transfer**) has administrative controls that support protocols like SFTP and FTPS. Used in the banking

industry, MFT provides additional encryption during financial file transfers.

- **What tasks can be automated in infrastructure management?**
- Applying automation to common management tasks—like provisioning, configuring, deploying, and decommissioning—simplifies operations at scale, allowing you to regain control over and visibility into your infrastructure. What IT infrastructure processes can be automated?



What is ECR in ECS?

ECR is an acronym for the “Elastic container registry”. In simple terms, it is the so-called “AMAZON DOCKER HUB” of your containers! You can push all your local containers images from your LOCAL to ECR, ECR is the home for all your pushed container images where later it can be used by ECS service to get deployed on AMAZON platform!