



EVERYMOBILE – ANDROID INSTALLATION MANUAL



Contact:

sales@testonneed.com

Follow us:

[LinkedIn](#) and [Twitter](#)

Revisions

Version	Primary Author(s)	Description of Version	Date Completed
0.1	Sahana Badri	First Draft version	

Review & Approval

Requirements Document Approval History

Approving Party	Version Approved	Signature	Date

Requirements Document Review History

Reviewer	Version Reviewed	Signature	Date

Table of Contents

1. Software To Be Installed	3
2. Installation – Notepad ++	4
3. Installation – Python.....	7
3.1 Setting up Environmental Variables.....	9
3.2 Install – WxPython.....	12
3.3 Install – Setup Tools	15
3.4 Install – PIP	16
4. Installation – Robot Framework and RIDE	17
4.1 Installation – Robot Framework	17
4.2 Installation – RIDE	18
5. Installation – ELK	20
6. Installation – Jenkins.....	23
6.1 Creating New view	31
6.2 Creating New Job	33
7. Installation – Node	36
7.1 Setting Environmental Variable	41
8. Installation – Appium.....	43
9. Installation – Android Studio	45
9.1 Setting Up Environmental Variables	58
10. Installation – Genymotion.....	62
11. Setting The Phone	76

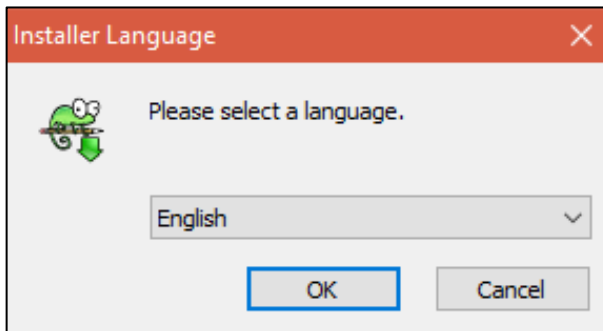
1. SOFTWARE TO BE INSTALLED

The software to be installed are listed below. The process of installation is also explained further in the document.

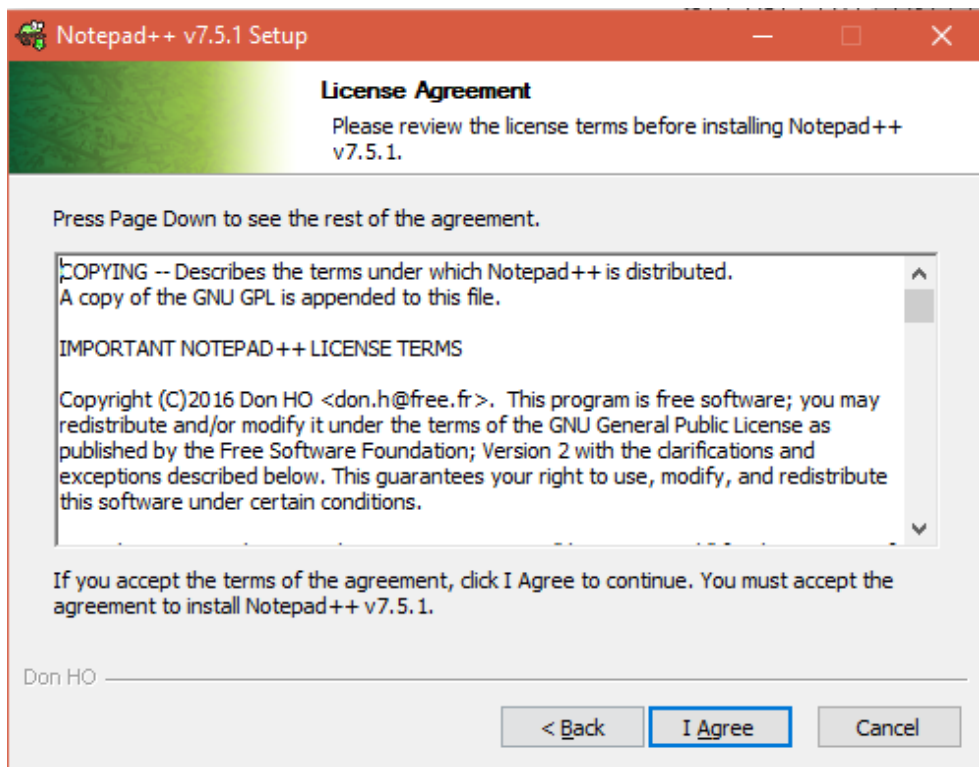
- Notepad ++
- Newman
- Python and PIP
- Robot Framework and RIDE
- ELK (Elastic Search, Logstash, Kibana)
- Jenkins
- Selenium
- Chrome driver
- Node
- Appium
- Android Studio
- Genymotion
- Android Phone

2. INSTALLATION – NOTEPAD ++

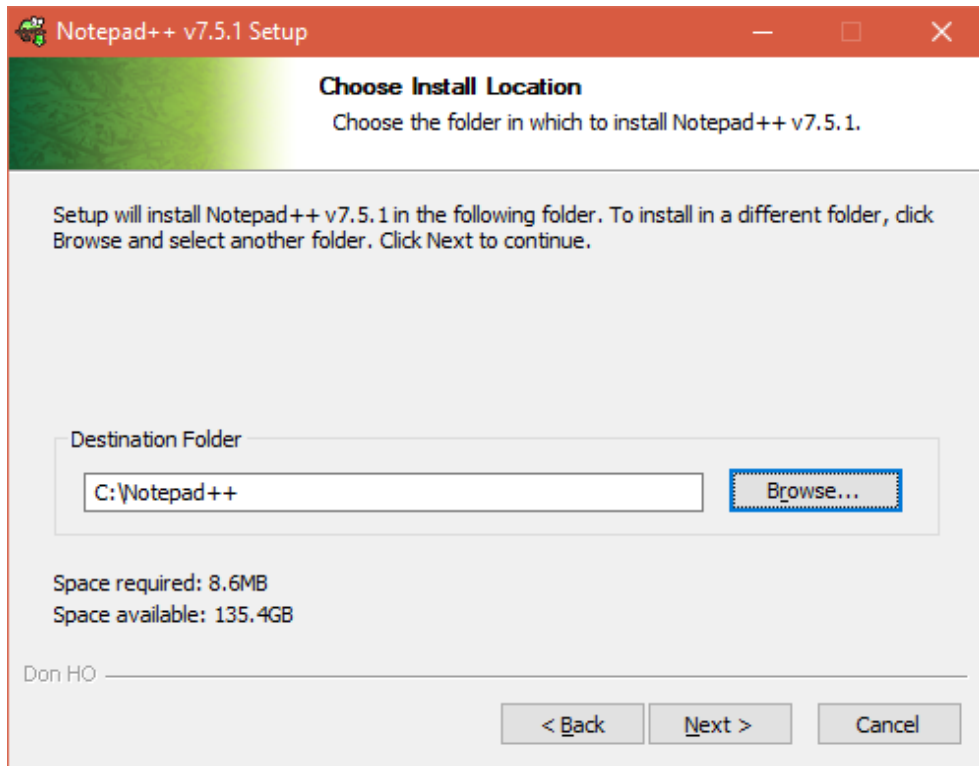
1. Download the notepad ++ file based on the OS requirements.
2. Run the installer.
3. It will ask to select a language.



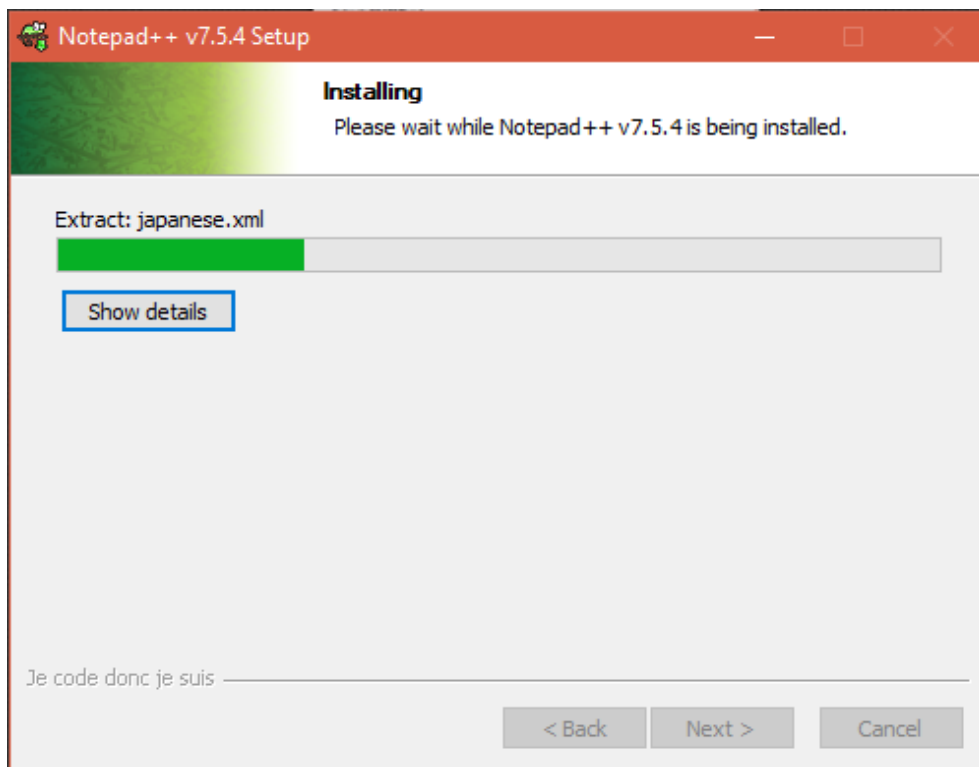
4. Accept the license agreement by clicking the **I Agree** button.



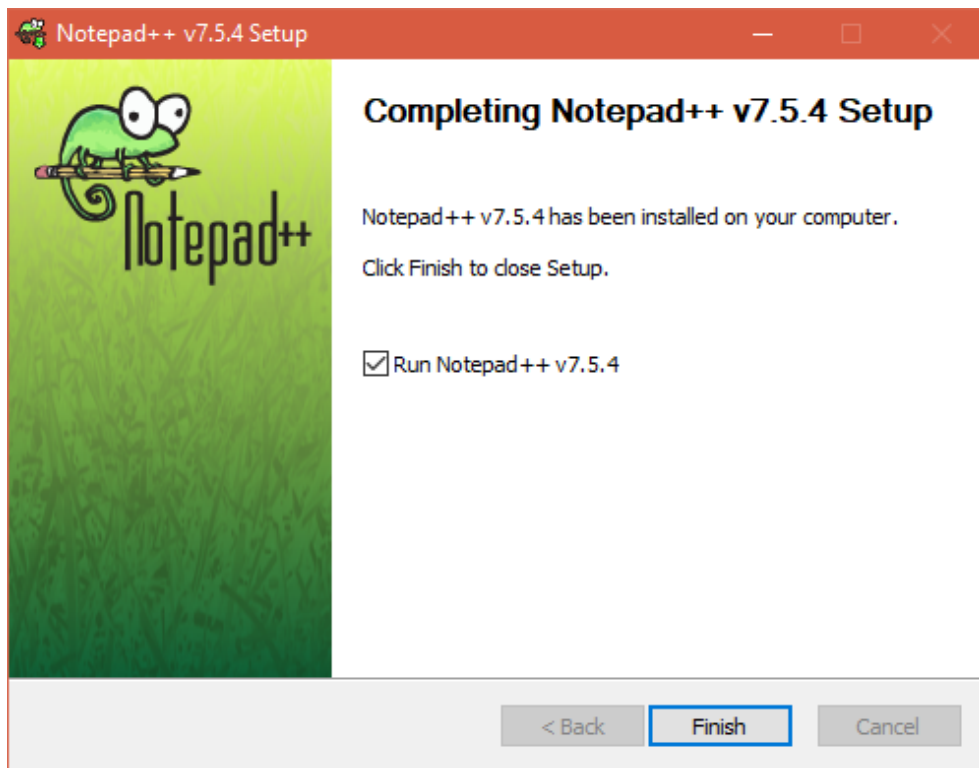
5. Change the folder to **C: drive** as shown below.



6. After setting the folder location, click Next button.
7. It will start to install.



8. On completing the installation, click Finish to end the setup.

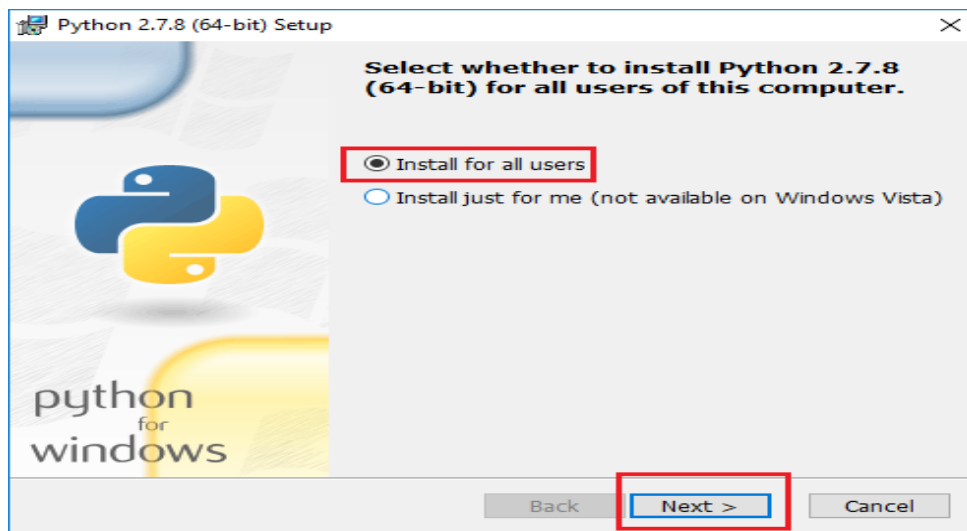


3. INSTALLATION – PYTHON

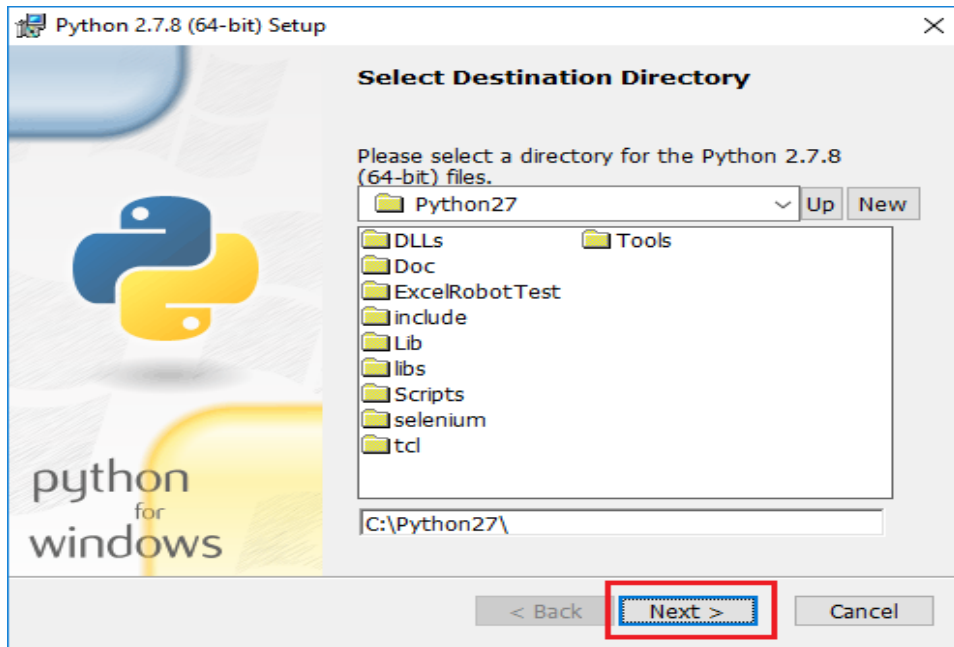
1. Download the Python 2.7.8 installer from <https://www.python.org/download/releases/2.7.8/>



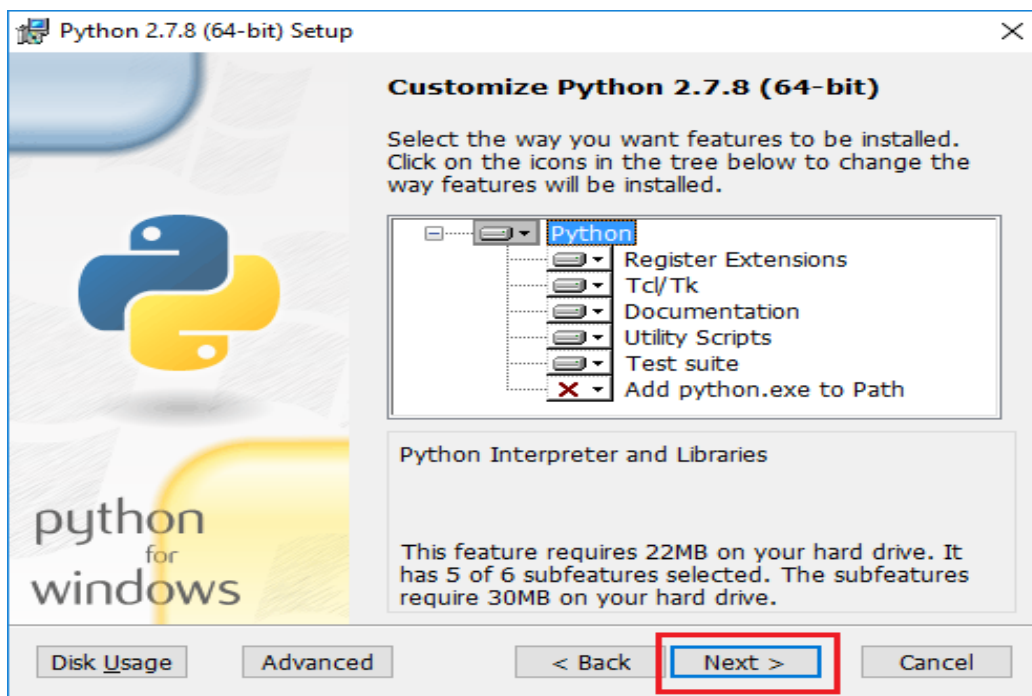
2. Run the installer, select “Install for all users,” and then click “**Next.**” The installer will even set the path variable.



3. On the directory selection screen, keep the installation directory as “C:\Python27\” and click “**Next.**”



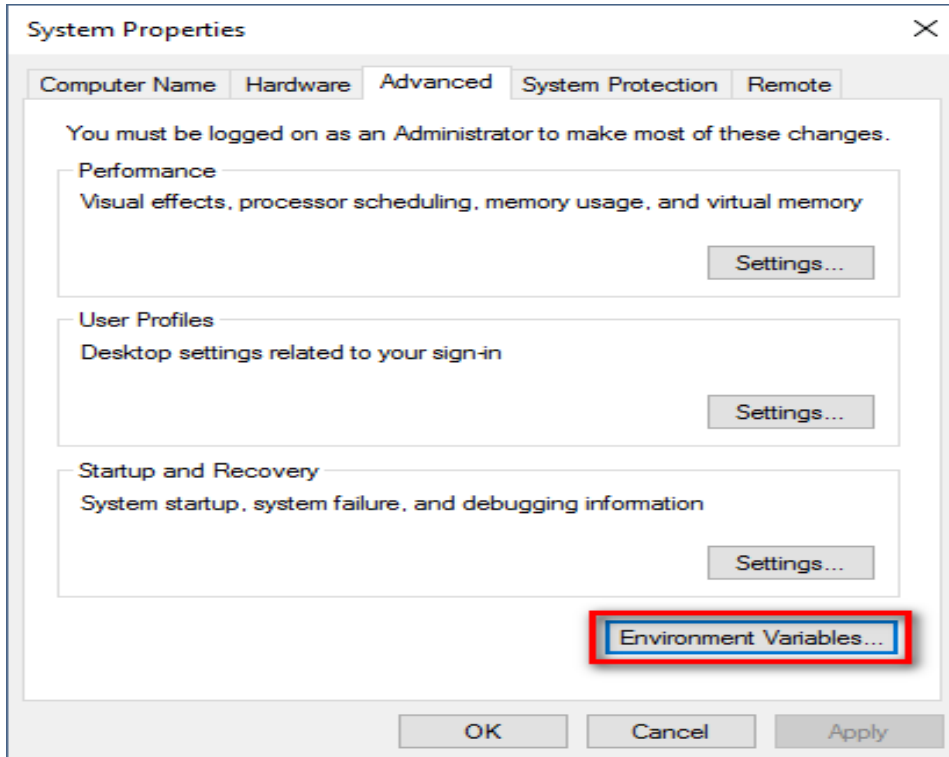
- On the customization screen, scroll down, to check the options for installing and then click **"Next"**.



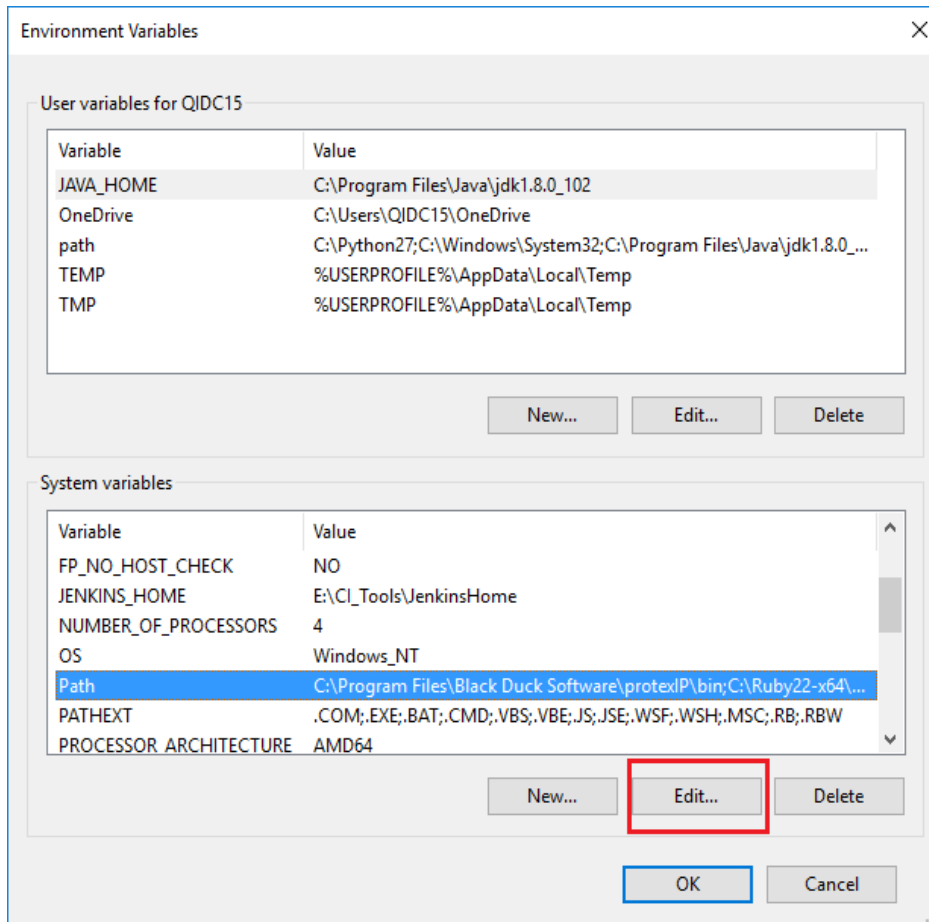
- Then click **"Finish"** once the installation is complete.

3.1 SETTING UP ENVIRONMENTAL VARIABLES

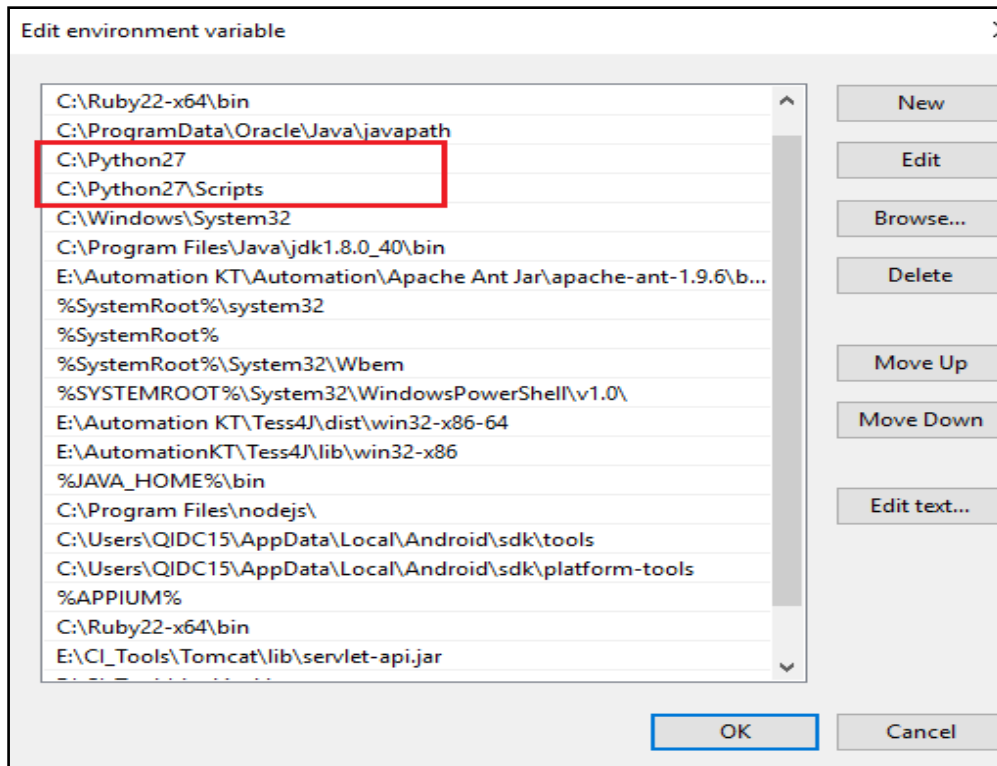
1. Hit Start, type “advanced system settings,” and then select the “View advanced system settings” option. In the “System Properties” window that opens, on the “Advanced” tab, click the “Environment Variables” button.
2. Open Start > Settings > Control Panel > System > Advanced > Environment Variables. There are User variables and System variables and the difference between them is that User variables affect only the current users, whereas System variables affect all users.



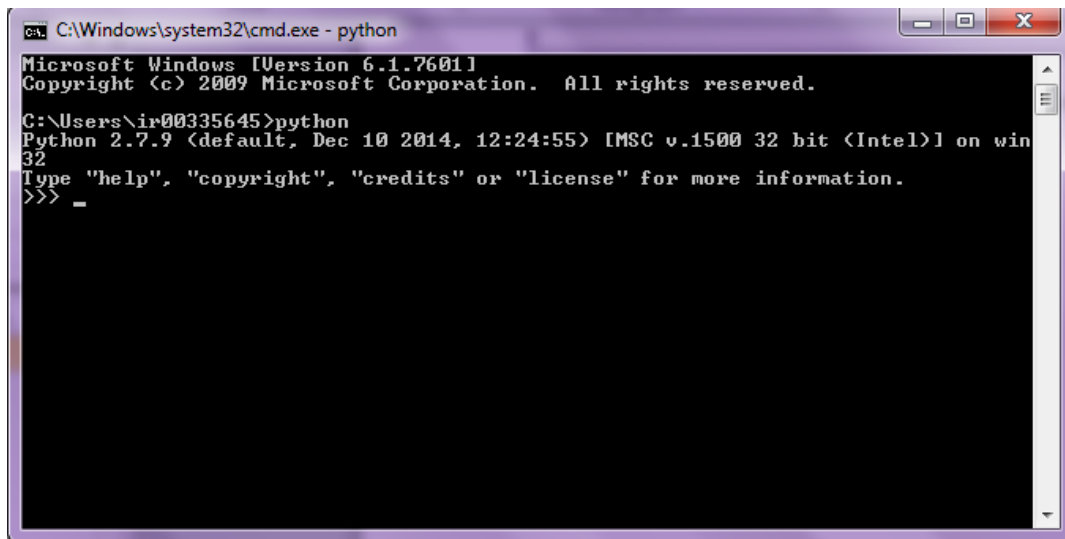
3. Find the “**Path**” variable under “**System Variables**” and select it.
4. After selection, click “**Edit**”. A window will open for where the path variables will be listed. Adding new path variable and editing existing variables can be done.



- Click **"New"** button and enter data in the format
Enter **"C:\Python27"**.
Again click **"New"** and enter **"C:\Python27\Scripts"**.
Click **"Ok"**

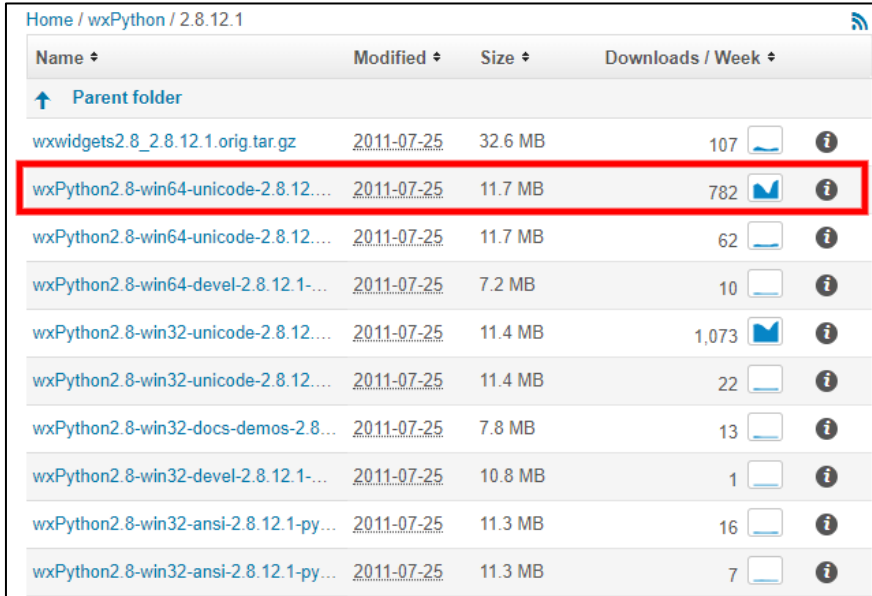


6. From command prompt, type **python -version** to make sure that Python has been installed properly.



3.2 INSTALL – WXPYTHON

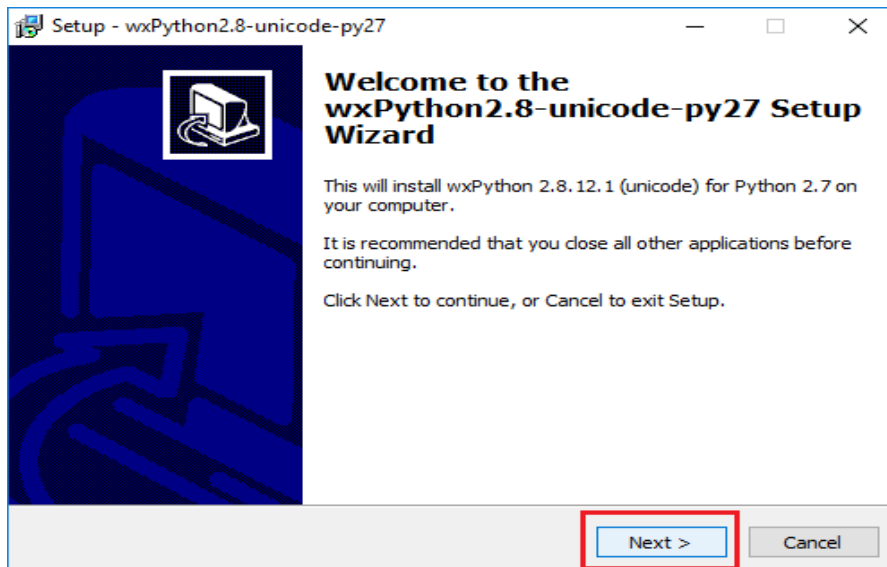
1. Download the wxPython installer from <https://sourceforge.net/projects/wxpython/files/wxPython/2.8.12.1/>
2. Open the link and click on “wxPython2.8-win64-unicode-2.8.12.1-py27.exe”



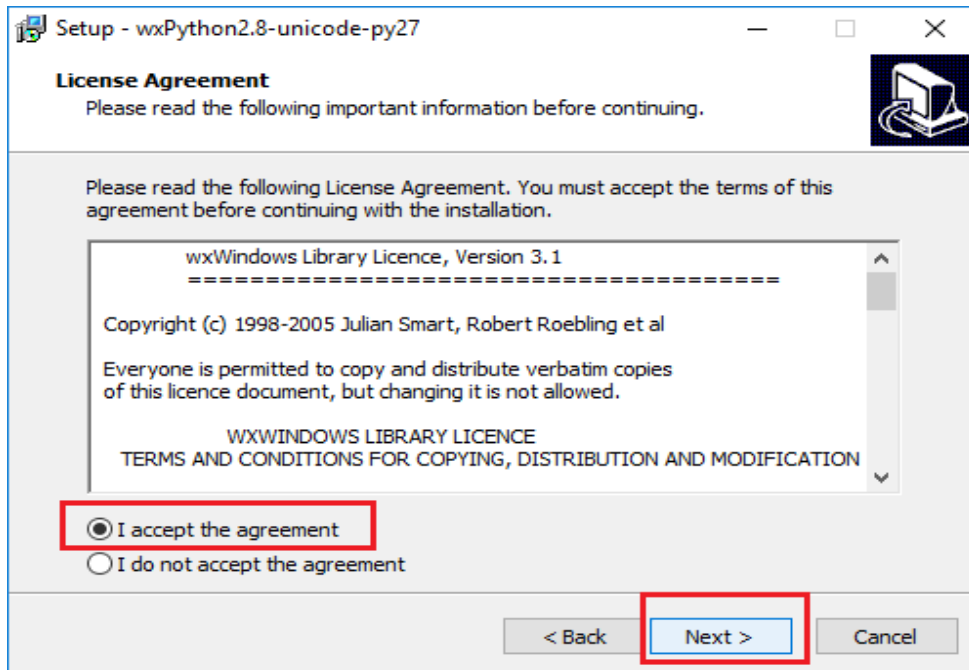
Home / wxPython / 2.8.12.1

Name ▾	Modified ▾	Size ▾	Downloads / Week ▾	
Parent folder				
wxwidgets2.8_2.8.12.1.orig.tar.gz	2011-07-25	32.6 MB	107	
wxPython2.8-win64-unicode-2.8.12.1-py27.exe	2011-07-25	11.7 MB	782	
wxPython2.8-win64-unicode-2.8.12.1-py31.exe	2011-07-25	11.7 MB	62	
wxPython2.8-win64-devel-2.8.12.1-py27.exe	2011-07-25	7.2 MB	10	
wxPython2.8-win32-unicode-2.8.12.1-py27.exe	2011-07-25	11.4 MB	1,073	
wxPython2.8-win32-unicode-2.8.12.1-py31.exe	2011-07-25	11.4 MB	22	
wxPython2.8-win32-docs-demos-2.8.12.1-py27.exe	2011-07-25	7.8 MB	13	
wxPython2.8-win32-devel-2.8.12.1-py27.exe	2011-07-25	10.8 MB	1	
wxPython2.8-win32-ansi-2.8.12.1-py27.exe	2011-07-25	11.3 MB	16	
wxPython2.8-win32-ansi-2.8.12.1-py31.exe	2011-07-25	11.3 MB	7	

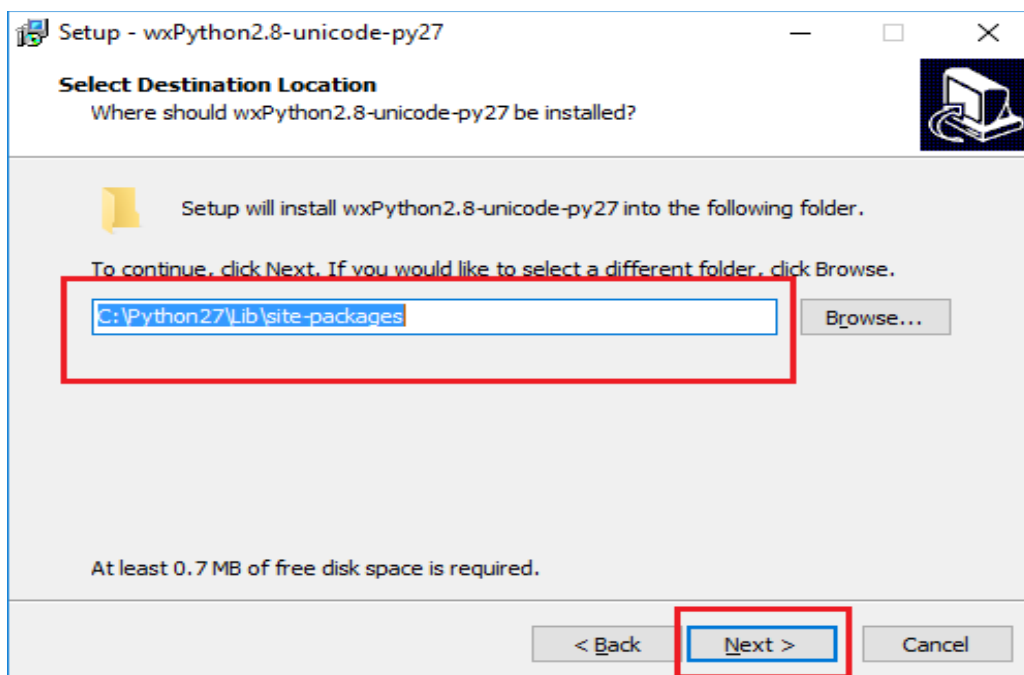
3. Run the installer and click “Next” to continue the setup.



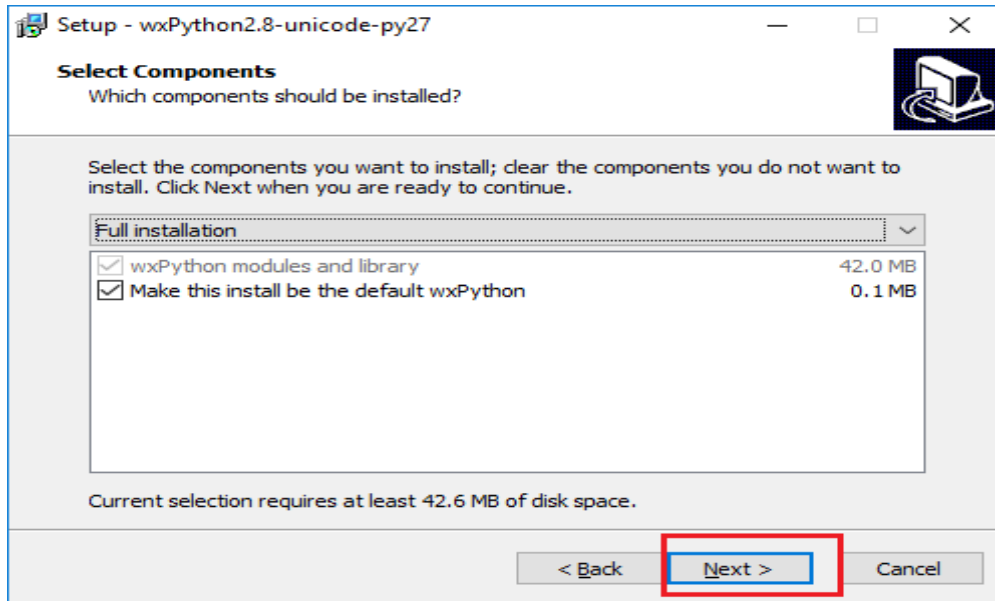
4. In the window, click on radio button to accept the terms of agreement.



5. Click “**Next**” after keeping the default directory “**C:\Python27\Lib\site-packages**” chosen for the installation for this program.



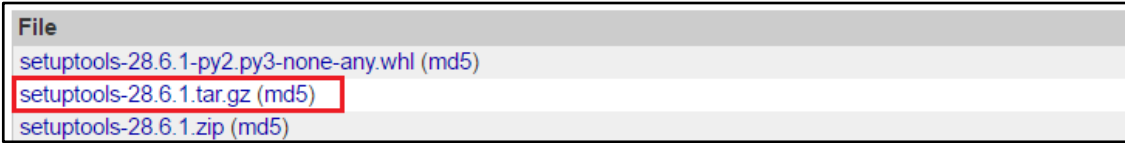
6. Click on “Next” after selecting “**Full Installation**” from the drop down and selecting the checkbox for “**Make this install be the default wxPython**”.



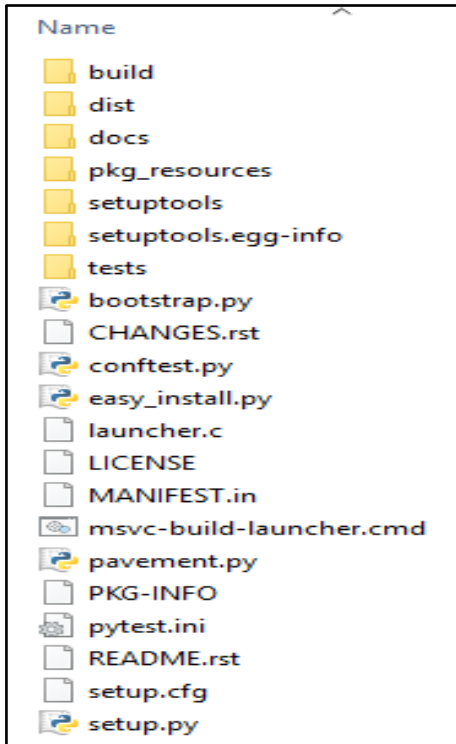
7. The installation process will follow and complete after this.
A command line interface will open and the installation will continue.
If the “**ReadMe.txt**” file is open, then close it.

3.3 INSTALL – SETUP TOOLS

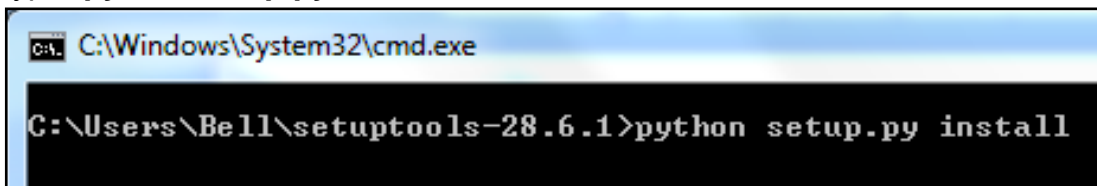
- Download setuptools from <https://pypi.python.org/pypi/setuptools/28.6.1>.
- Scroll down and click the link to download the setuptools zipped folder.



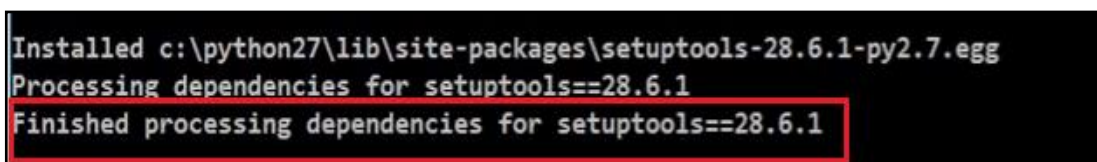
- Unzip the download folder in “In\Desired\Location” and the extracted folder “In\Desired\Location \setuptools-28.6.1” should have the contents as follows:



- Open the command line and browse by changing directory to the extracted folder and type “python setup.py install”



- The entire setup will run and the message will appear which will show the successful installation of setuptools.

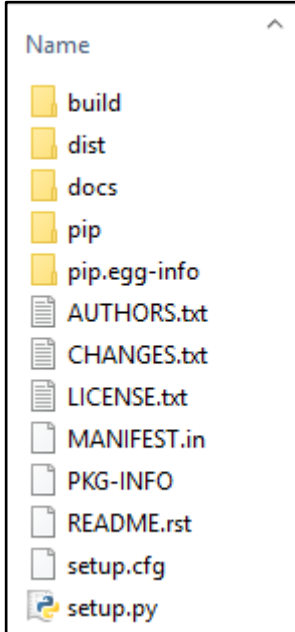


3.4 INSTALL – PIP

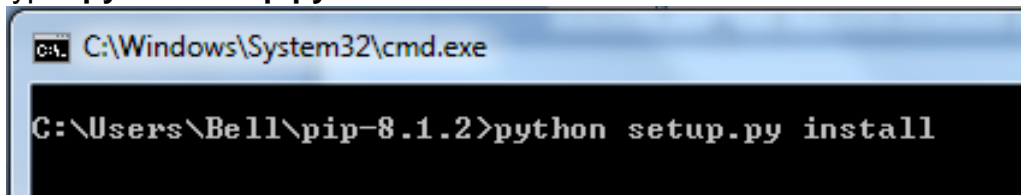
- Download pip from <https://pypi.python.org/pypi/pip/>
- Scroll down and click the link to download the pip zipped folder.



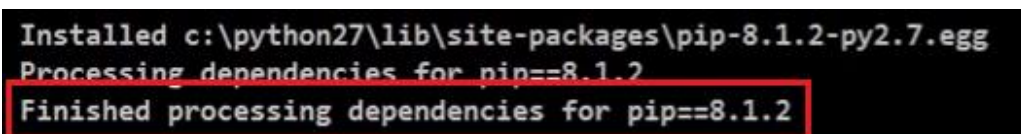
- Unzip the download folder in “In\Desired\Location” and the extracted folder “In\Desired\Location \pip-8.1.2” should have the contents as follows:



- Open the command line and browse by changing directory to the extracted folder and type “python setup.py install”.



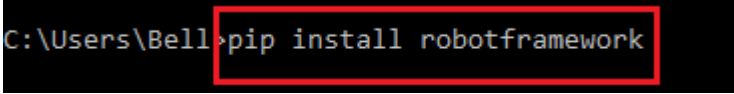
- The entire setup will run and the message will appear which will show the successful installation of pip.



4. INSTALLATION – ROBOT FRAMEWORK AND RIDE

4.1 INSTALLATION – ROBOT FRAMEWORK

1. Open the command prompt and browse to “C:\Users\<System_Name>” and type in “**pip install robotframework**”



```
C:\Users\Bell>pip install robotframework
```

2. The successful message will be shown after installation.

4.2 INSTALLATION – RIDE

- Open the command prompt and browse to “C:\Users\<System_Name>” and type in “**pip install robotframework-ride**”

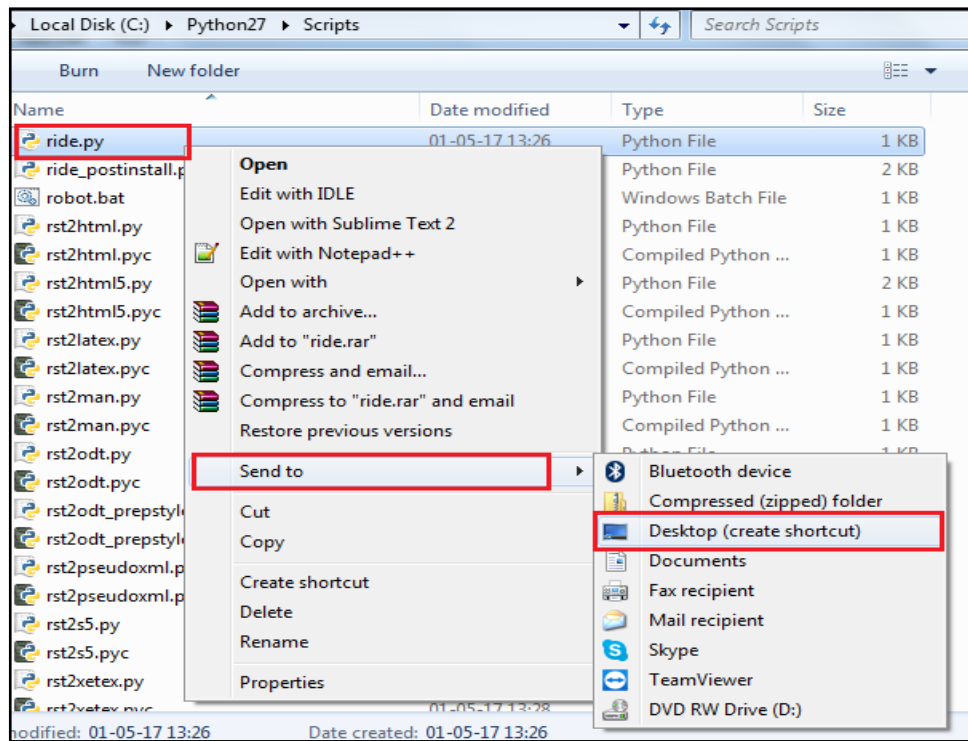
```
C:\Users\Bell>pip install robotframework-ride
```

- The successful message will be shown after installation.

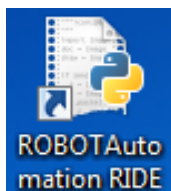
```
Installing collected packages: robotframework-ride
  Running setup.py install for robotframework-ride ... done
Successfully installed robotframework-ride-1.5.2.1
```

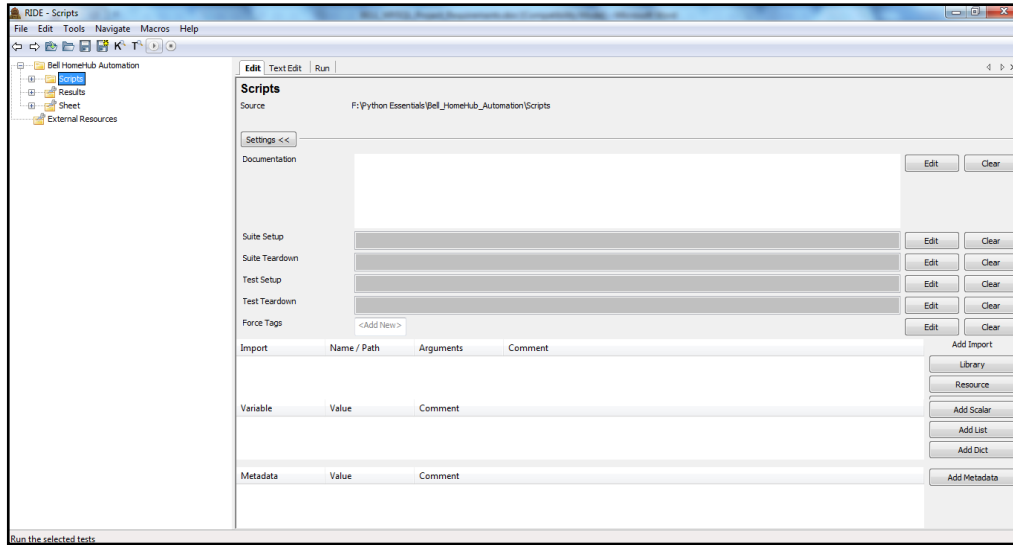
- Create a shortcut of RIDE on the desktop by opening the folder “C:\Python27\Scripts” and right-click on “**ride.py**”.

After the popup menu appears, select “**Send to**” and select “**Desktop (create shortcut)**”



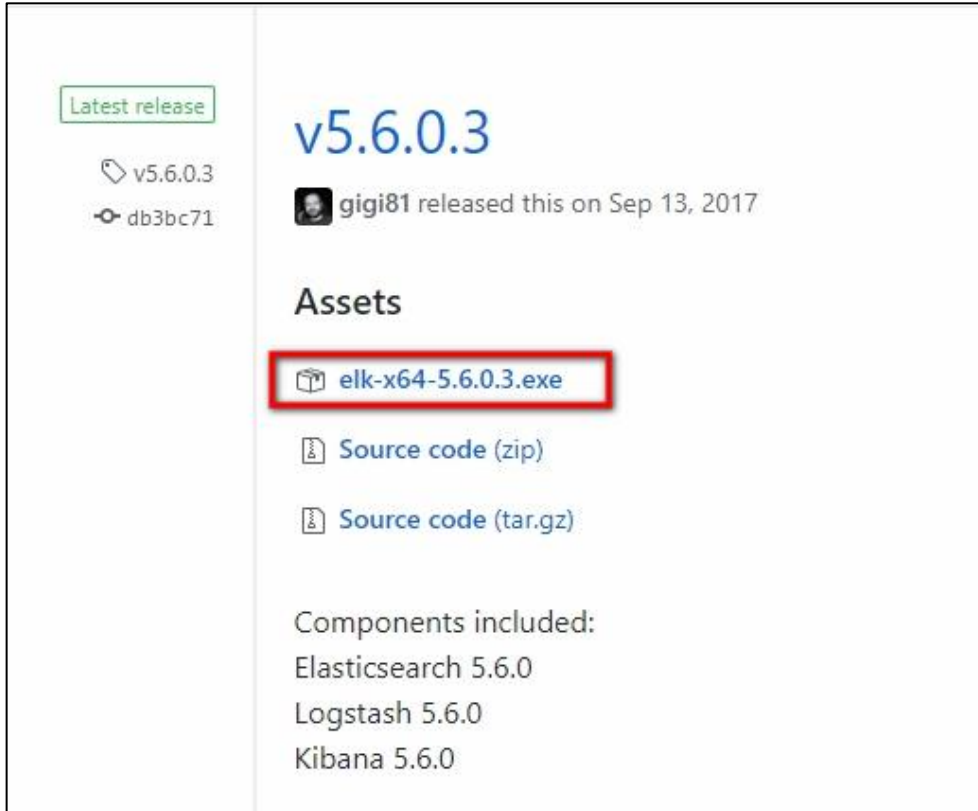
- Rename the created shortcut to “**ROBOTAutomation RIDE.**”
- Double click on shortcut icon to open the RIDE window.



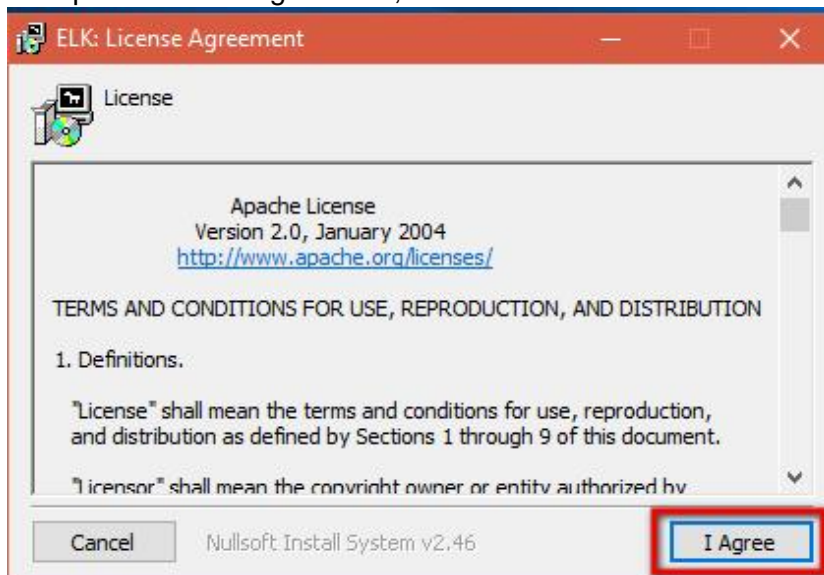


5. INSTALLATION – ELK

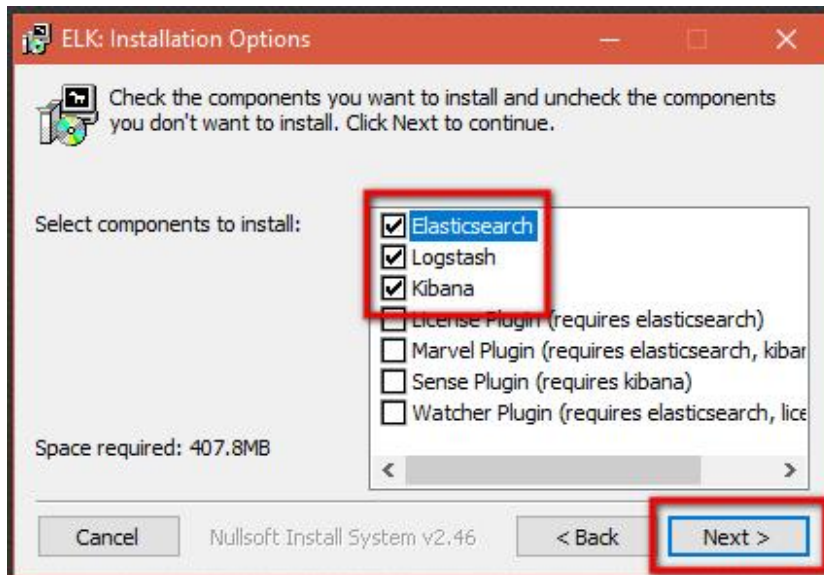
- Click the link below to download the ELK (Elastic Search, Kibana and Logstash) installer:
<https://www.codeproject.com/Tips/1083311/ELK-Stack-Install-on-Windows>
- In the browser, link the installer to start the download.



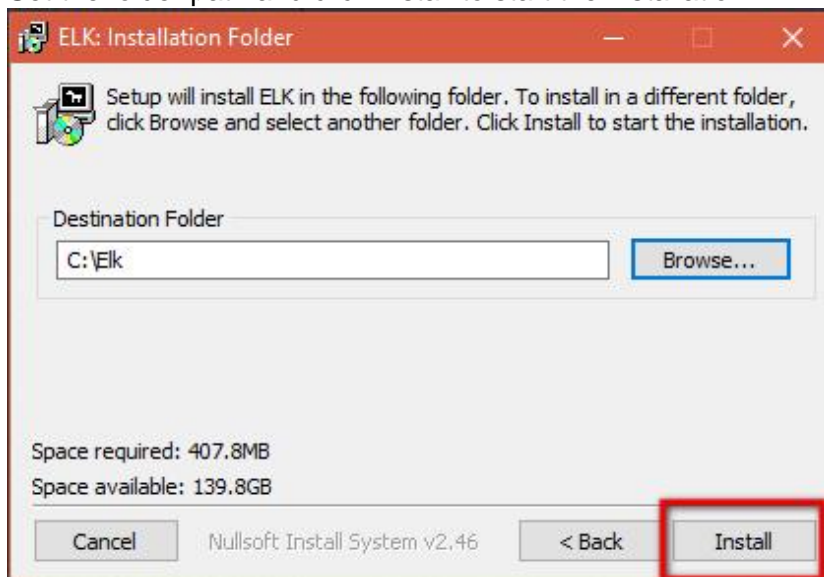
- Run the installer from the downloads.
- Accept the license agreement, and click continue to install.



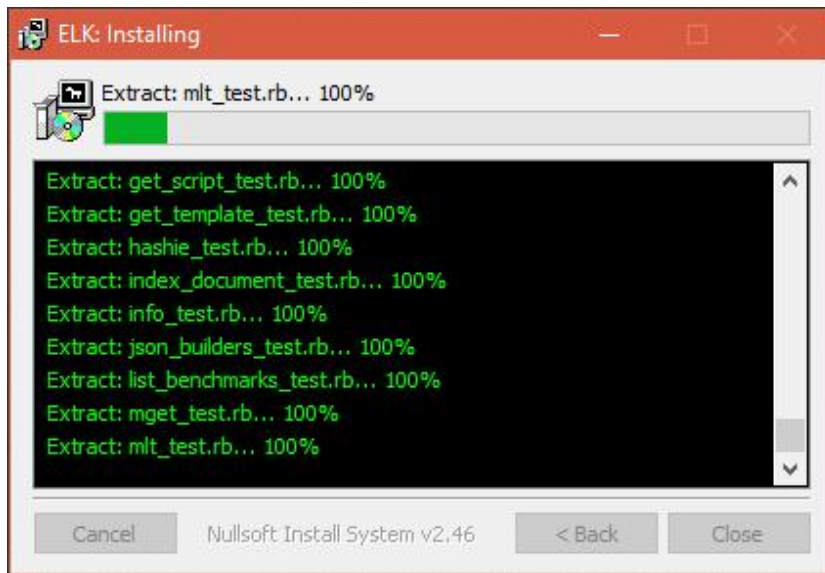
- e. Select Elasticsearch, Logstash and Kibana and click Next button.



- f. Set the folder path and click install to start the installation.



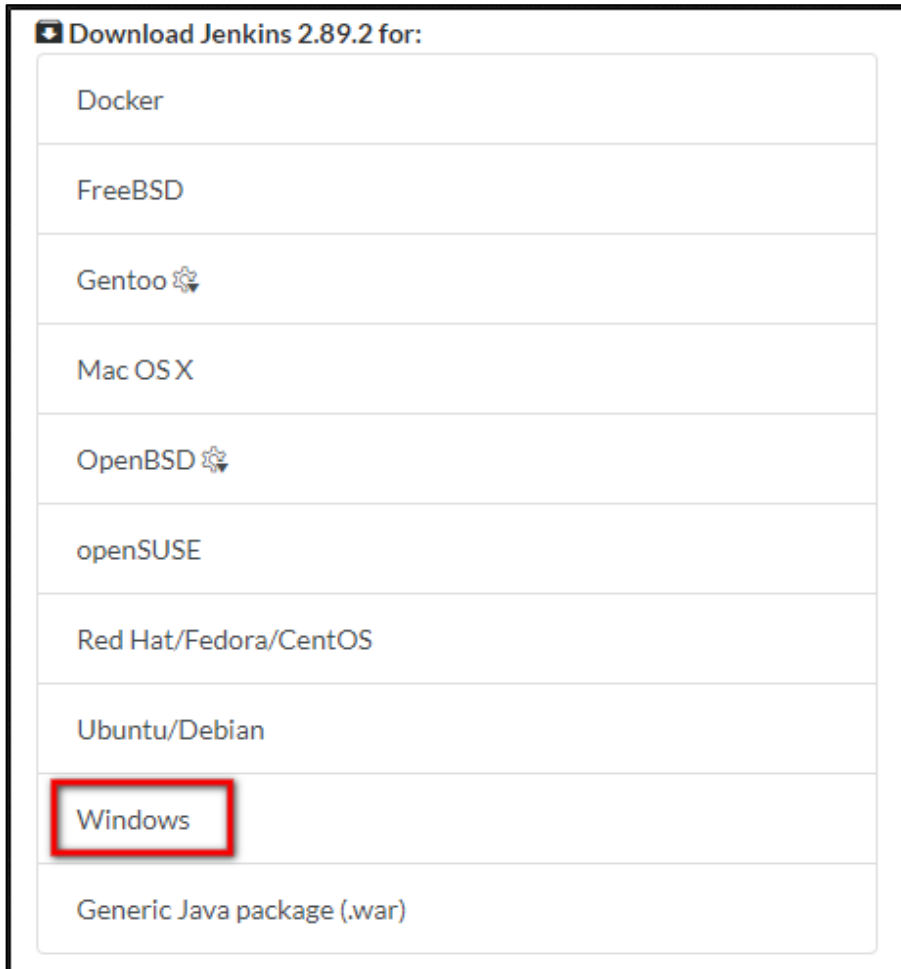
- g. It will start the installation.



h. Once the installation is complete, click Finish to end the setup.

6. INSTALLATION – JENKINS

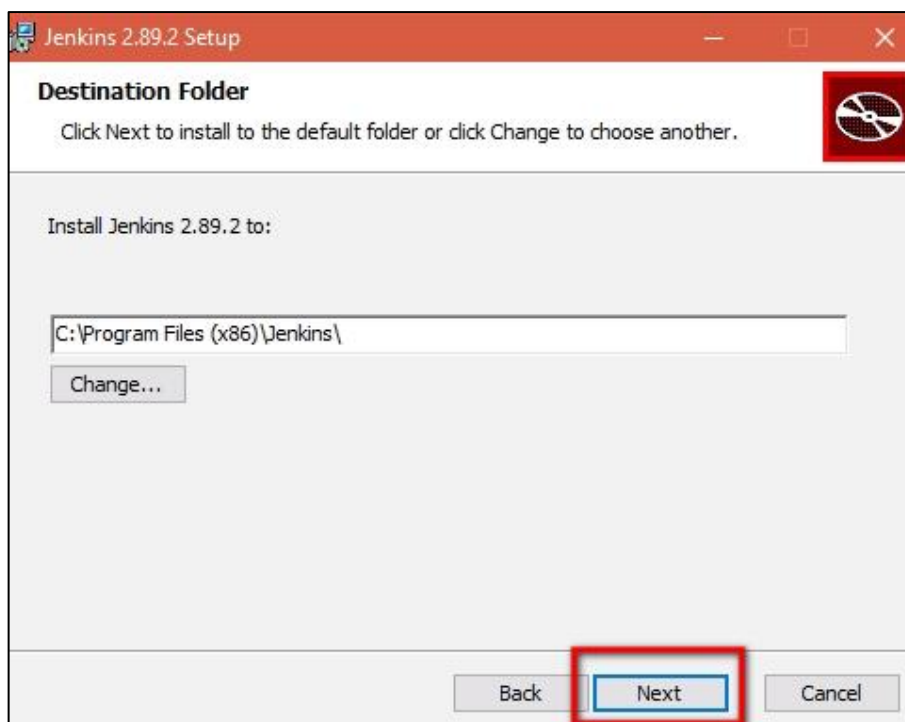
1. Click on the link below to download Jenkins:
<https://jenkins.io/download/>
2. Scroll down and select to download based on the OS requirement. In the below example, it is shown for windows.



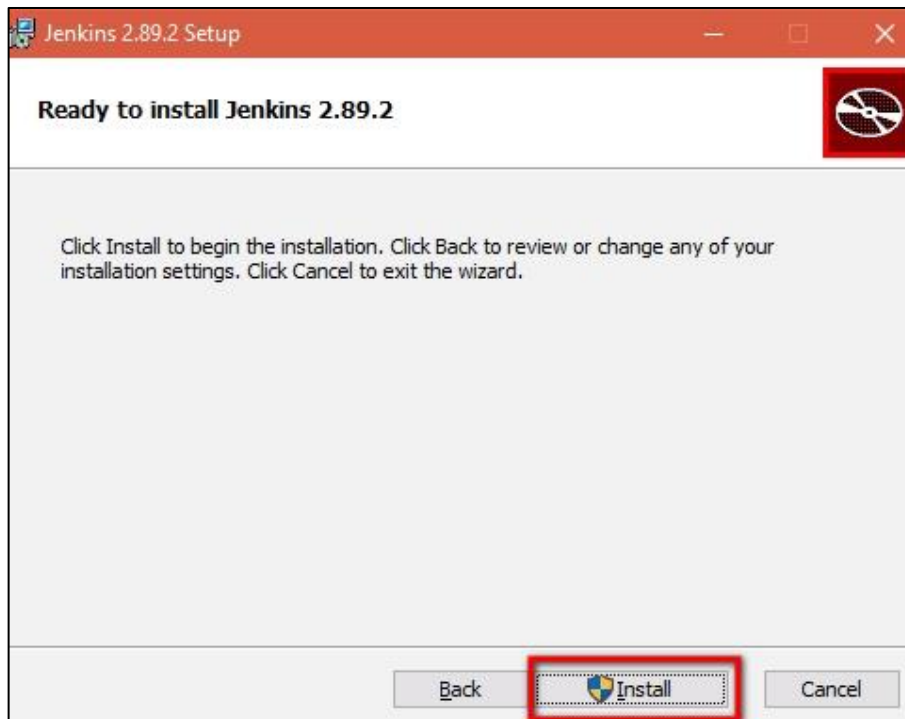
3. Run the installer.
4. Click **Next** button to accept the license agreement.



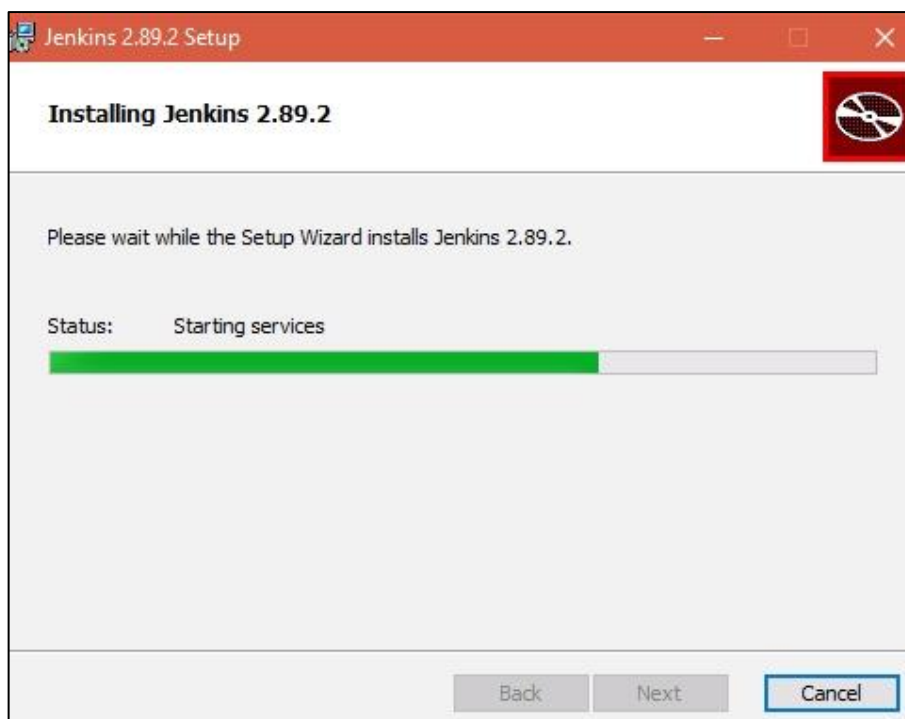
5. click **Next** button to set the folder location.



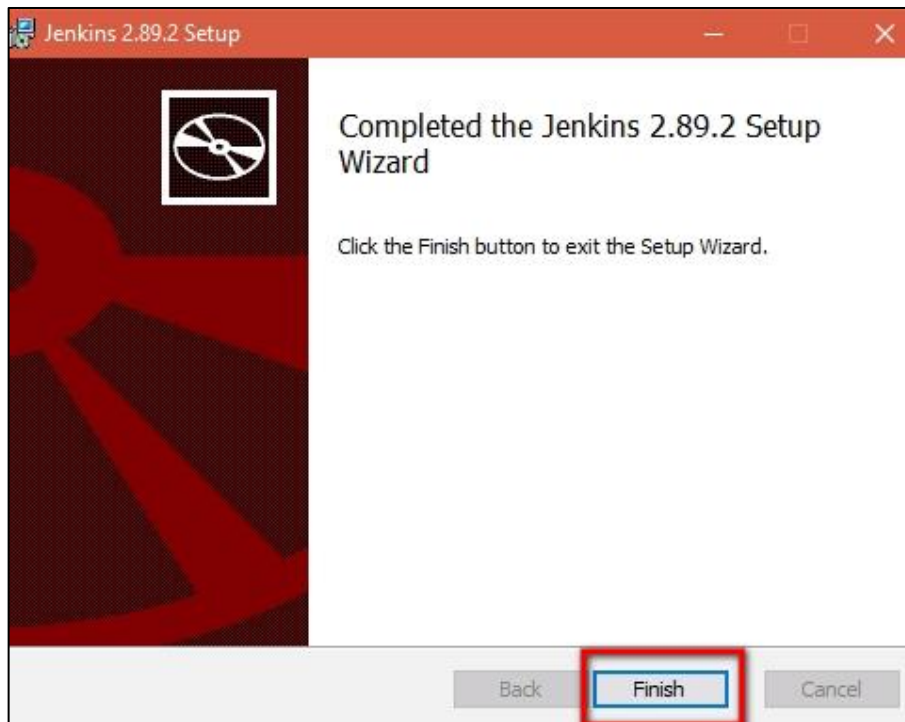
6. Click Install to start the installation.



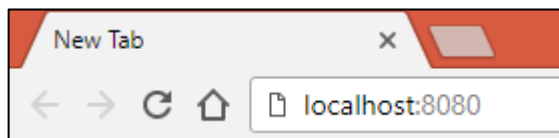
7. Then it will start to install.



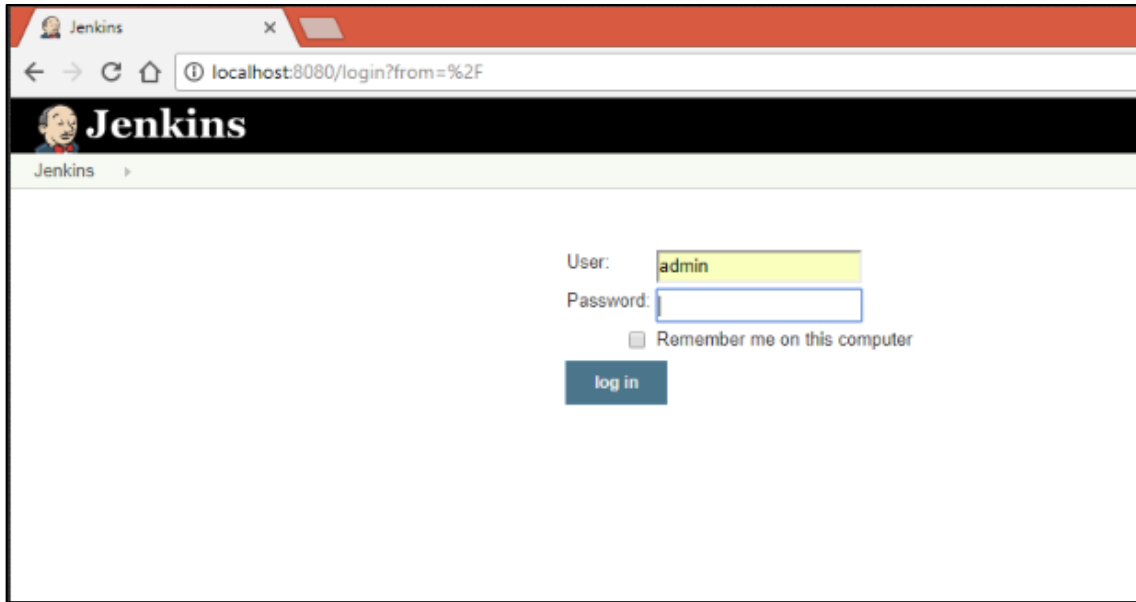
8. Once complete, click Finish to end the installation.



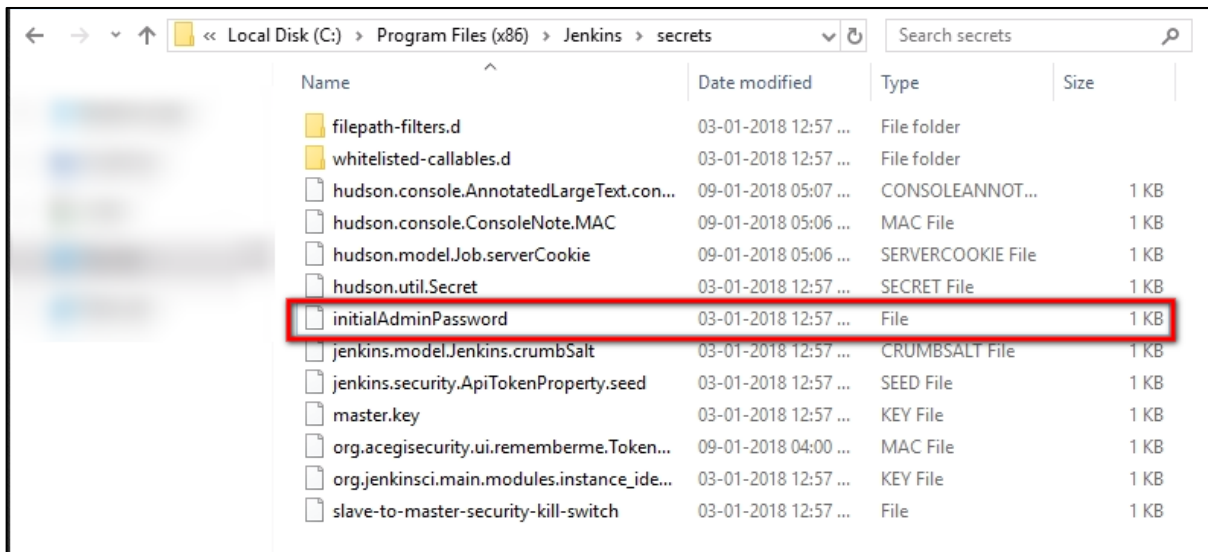
9. Open chrome
10. In the URL bar, enter "localhost:8080" and press enter



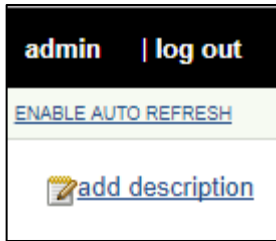
11. It will open Jenkins login page.



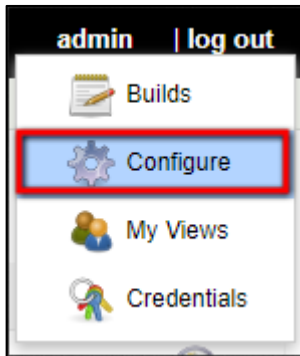
12. For the first to login, the password will be given in the following folder location:
C:\Program Files (x86)\Jenkins\secrets
13. In the above folder, find a file called **initialAdminPassword**.



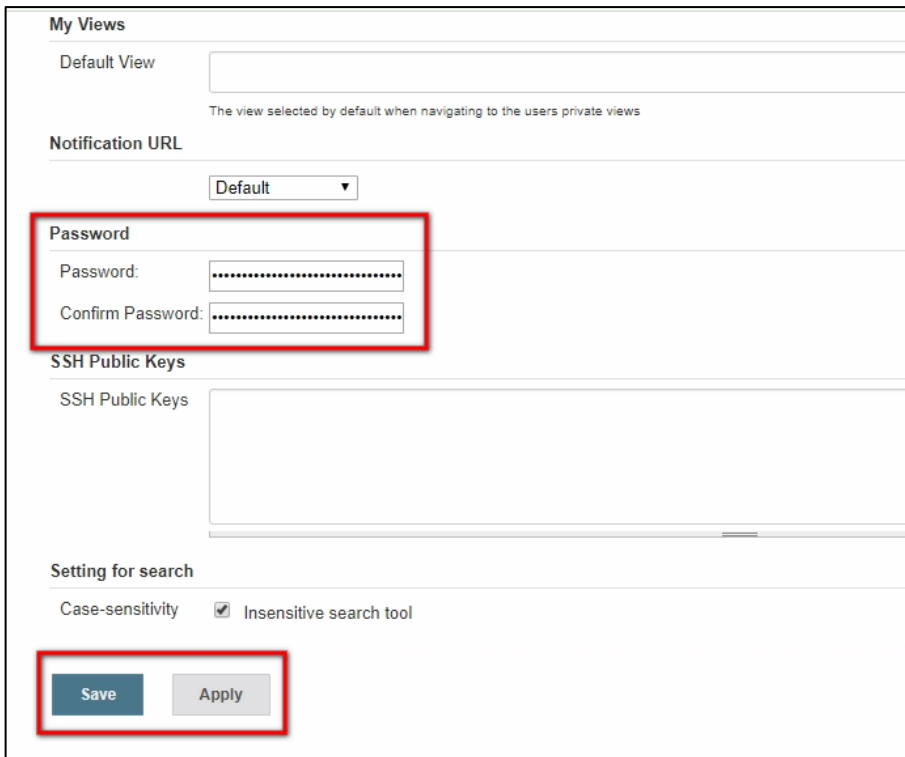
14. Open that file and copy the contents inside and paste the same in password text box.
15. After logging in, on the right top, find the user name that was given during logging in.



16. Under that click Configure.



17. Scroll down and find the Password and Confirm Password.



My Views

Default View

The view selected by default when navigating to the users private views

Notification URL

Default ▼

Password

Password:

Confirm Password:

SSH Public Keys

SSH Public Keys


Setting for search


Case-sensitivity ☒ Insensitive search tool


18. Now here change to the desired password. Click Apply and Save to save the changes.
19. To create a new job, click **New Item**.
20. Give the test case name to create a new job.


Enter an item name


» This field cannot be empty, please enter a valid name



Freestyle project
This is the central feature of Jenkins. Jenkins will build your project, combining any SCM with any build system, and this can be even used for something other than software build.


Pipeline
Orchestrates long-running activities that can span multiple build slaves. Suitable for building pipelines (formerly known as workflows) and/or organizing complex activities that do not easily fit in free-style job type.


Multi-configuration project
Suitable for projects that need a large number of different configurations, such as testing on multiple environments, platform-specific builds, etc.


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.


OK


21. Then select the type of the job that is to be created.
22. Click Ok to save the job.


Enter an item name


BL_BELL_Login


» Required field



Freestyle project
This is the central feature of Jenkins. Jenkins will build your project, combining any SCM with any build system, and this can be even used for something other than software build.


Pipeline
Orchestrates long-running activities that can span multiple build slaves. Suitable for building pipelines (formerly known as workflows) and/or organizing complex activities that do not easily fit in free-style job type.


Multi-configuration project
Suitable for projects that need a large number of different configurations, such as testing on multiple environments, platform-specific builds, etc.

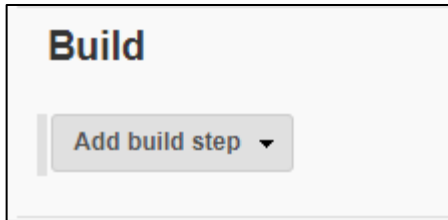

Folder
Creates a container that stores nested items in it. Useful for grouping things together. Unlike view, which is just a filter, a folder creates a separate namespace, so you can have multiple things of the same name as long as they are in different folders.


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

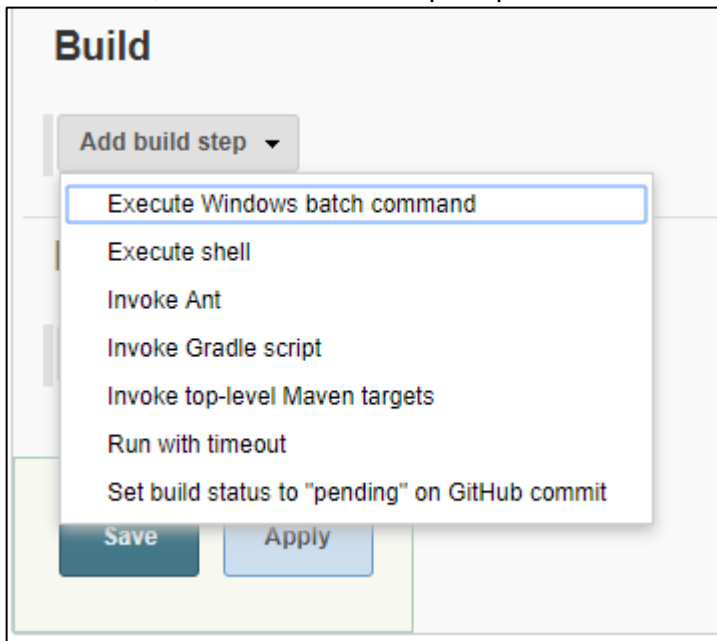

Multibranch Pipeline
Creates a set of Pipeline projects according to detected branches in one SCM repository.

OK

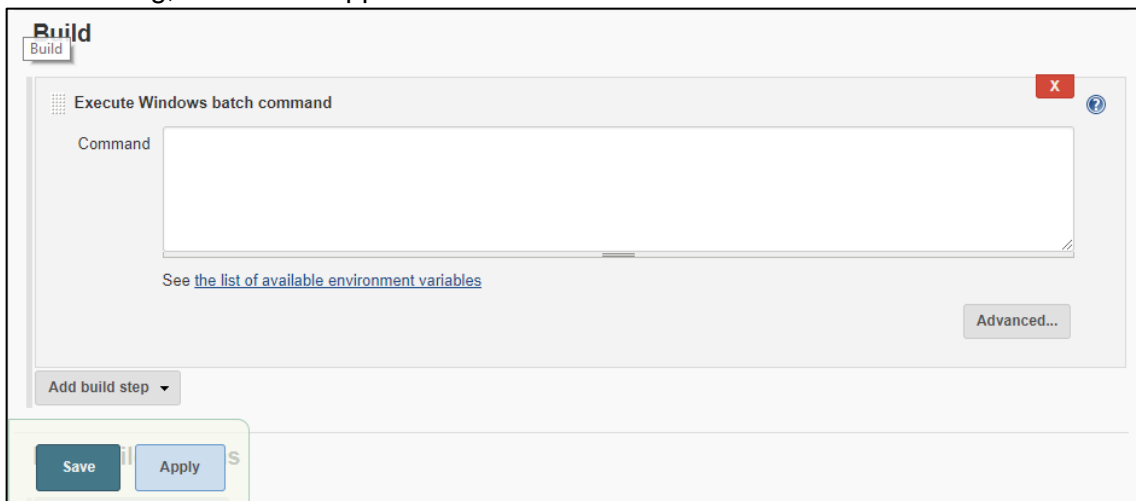
23. On click of Ok button, job configuration page will appear.
24. Scroll down and look for Build.



25. Under build, click Add Build Setup drop down.



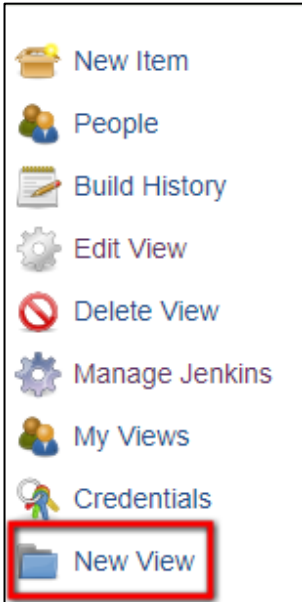
26. From the drop down, select **Execute Windows Batch Command**.
 27. On selecting, a text area appears.



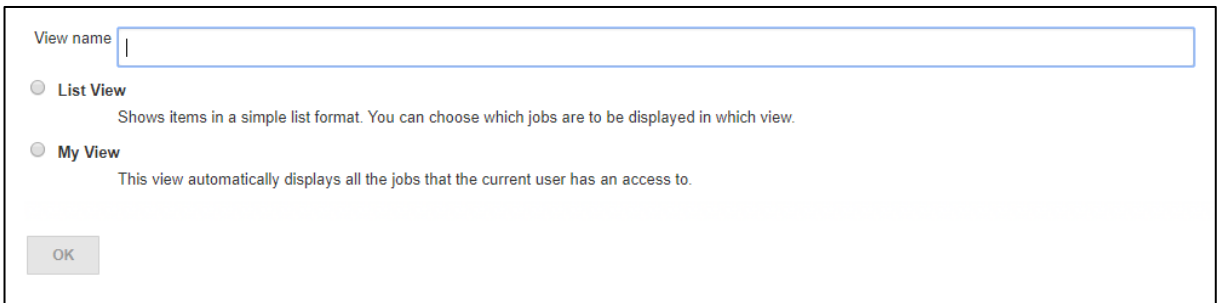
28. In the text area, write the following command.

6.1 CREATING NEW VIEW

1. To create a new view, on the left panel, click **New View**.

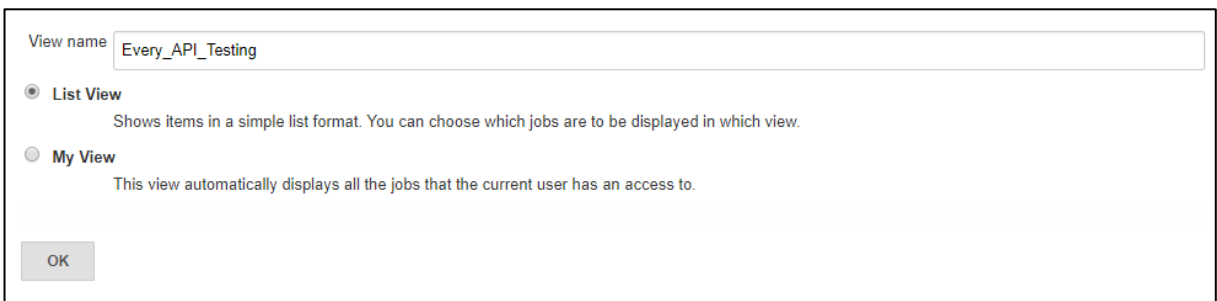


2. On clicking, give the name of the tab as **Every_Mobile_Testing** and select the type of view.



A screenshot of the 'New View' dialog box. It contains a text input field for 'View name' which is currently empty. Below the field are two radio button options: 'List View' and 'My View'. The 'List View' option is selected. Below the radio buttons are two lines of descriptive text: 'Shows items in a simple list format. You can choose which jobs are to be displayed in which view.' and 'This view automatically displays all the jobs that the current user has an access to.' At the bottom left is an 'OK' button.

3. On entering the name and selecting the view type, the OK button becomes active. Click Ok.



A screenshot of the 'New View' dialog box. The 'View name' field now contains the text 'Every_API_Testing'. The 'List View' radio button remains selected. The 'OK' button at the bottom left is now active (highlighted in grey).

4. In the next page, select the jobs that are to be added to this tab.

View name

Every_Mobile_Testing

☒ **List View**
Shows items in a simple list format. You can choose which jobs are to be displayed in which view.

☐ **My View**
This view automatically displays all the jobs that the current user has an access to.

OK

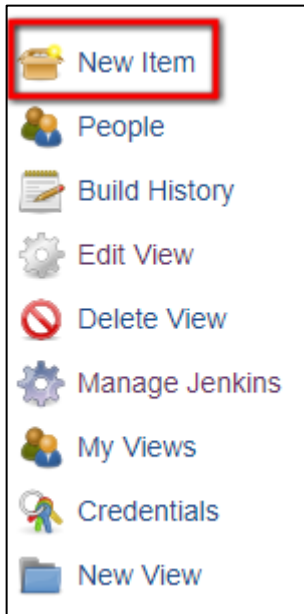
5. Once done, click Apply and Ok to save and Close the creation.

OK

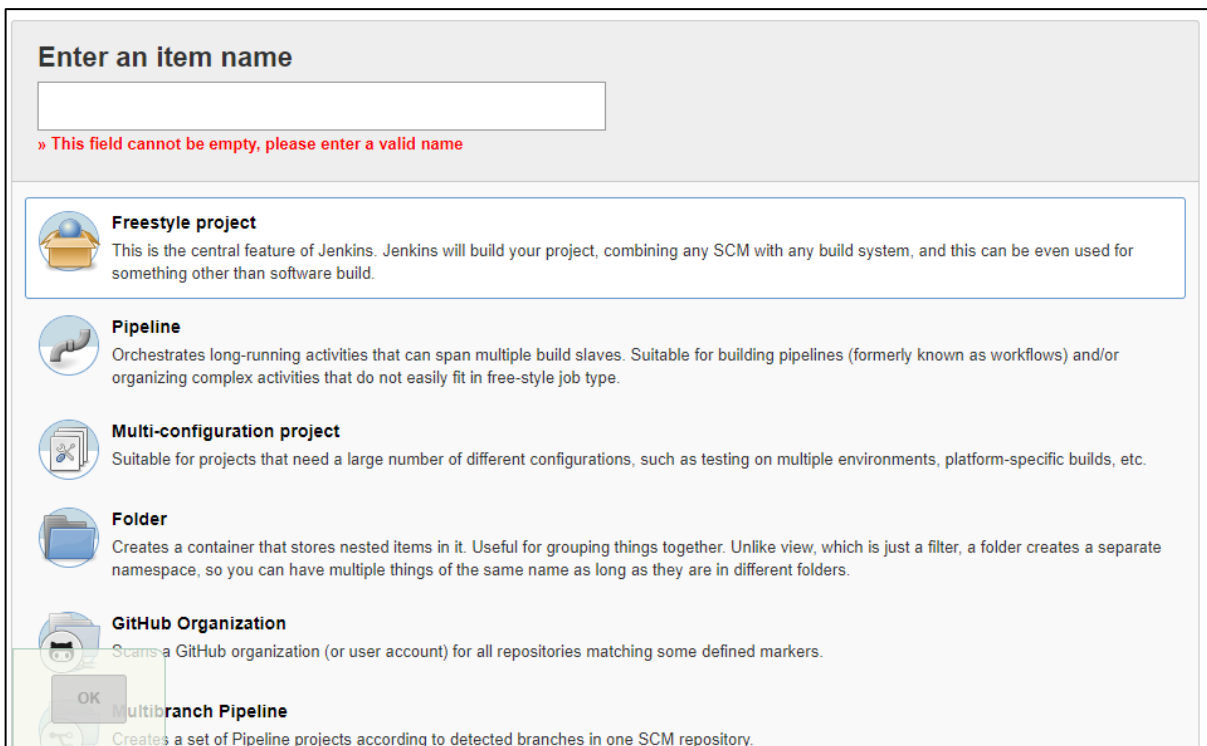
Apply

6.2 CREATING NEW JOB

1. To create a new job, on the left panel, click **New Item**.



2. Give a name for the new job that is going to be created and select the tab in which it is to be stored.



Enter an item name

» This field cannot be empty, please enter a valid name

- Freestyle project**
This is the central feature of Jenkins. Jenkins will build your project, combining any SCM with any build system, and this can be even used for something other than software build.
- Pipeline**
Orchestrates long-running activities that can span multiple build slaves. Suitable for building pipelines (formerly known as workflows) and/or organizing complex activities that do not easily fit in free-style job type.
- Multi-configuration project**
Suitable for projects that need a large number of different configurations, such as testing on multiple environments, platform-specific builds, etc.
- 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.


OK

3. Then select the type of the job that is to be created.
4. Click Ok to save the job.


Enter an item name

MF_Calc_001


» Required field

**Freestyle project**


This is the central feature of Jenkins. Jenkins will build your project, combining any SCM with any build system, and this can be even used for something other than software build.

**Pipeline**


Orchestrates long-running activities that can span multiple build slaves. Suitable for building pipelines (formerly known as workflows) and/or organizing complex activities that do not easily fit in free-style job type.

**Multi-configuration project**


Suitable for projects that need a large number of different configurations, such as testing on multiple environments, platform-specific builds, etc.

**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.

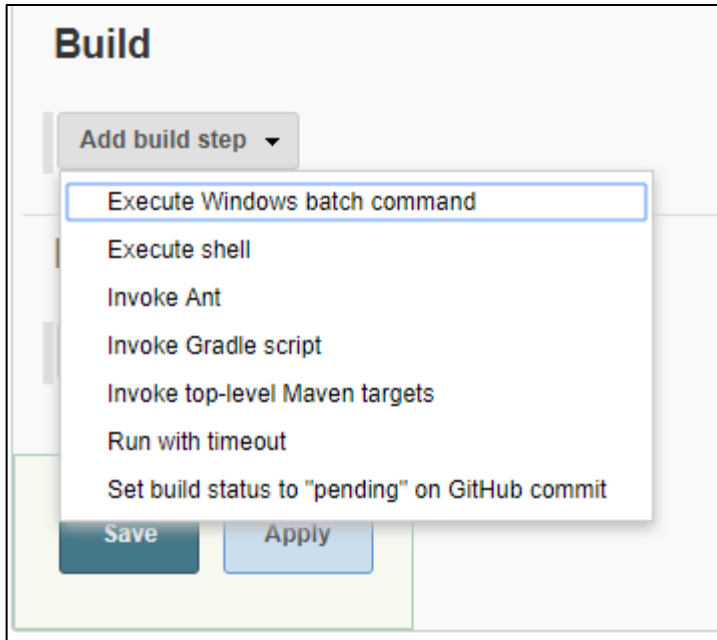
OK

- On click of Ok button, job configuration page will appear.
- Scroll down and look for Build.

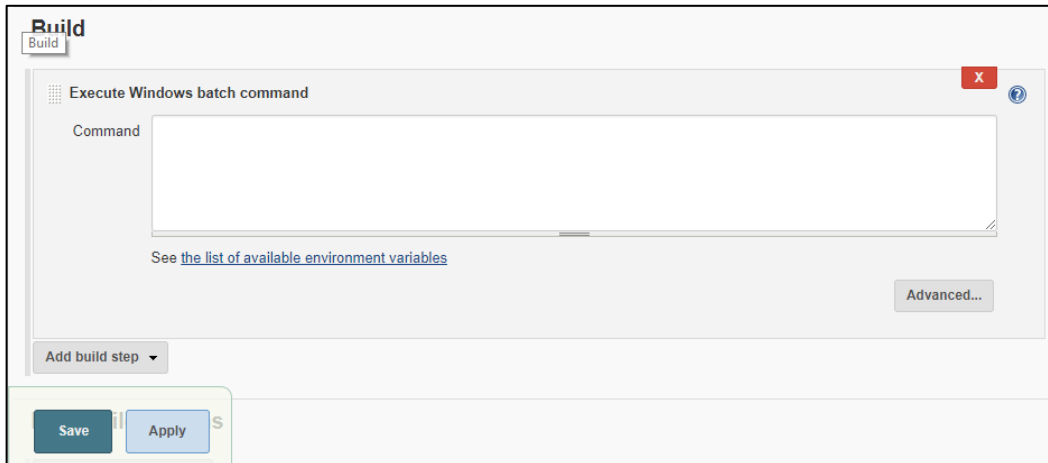
Build

Add build step ▼

- Under build, click Add Build Setup drop down.



8. From the drop down, select **Execute Windows Batch Command**.
9. On selecting, a text area appears.



10. In the text area, write the following command.

7. INSTALLATION – NODE

1. Click the link to download the node server
<https://nodejs.org/en/download/>
2. Look for the version of that support the user's OS like below: (Example shows for Windows)

LTS

Recommended For Most Users

Current

Latest Features



Windows Installer

node-v8.9.3-x86.msi



Macintosh Installer

node-v8.9.3.pkg

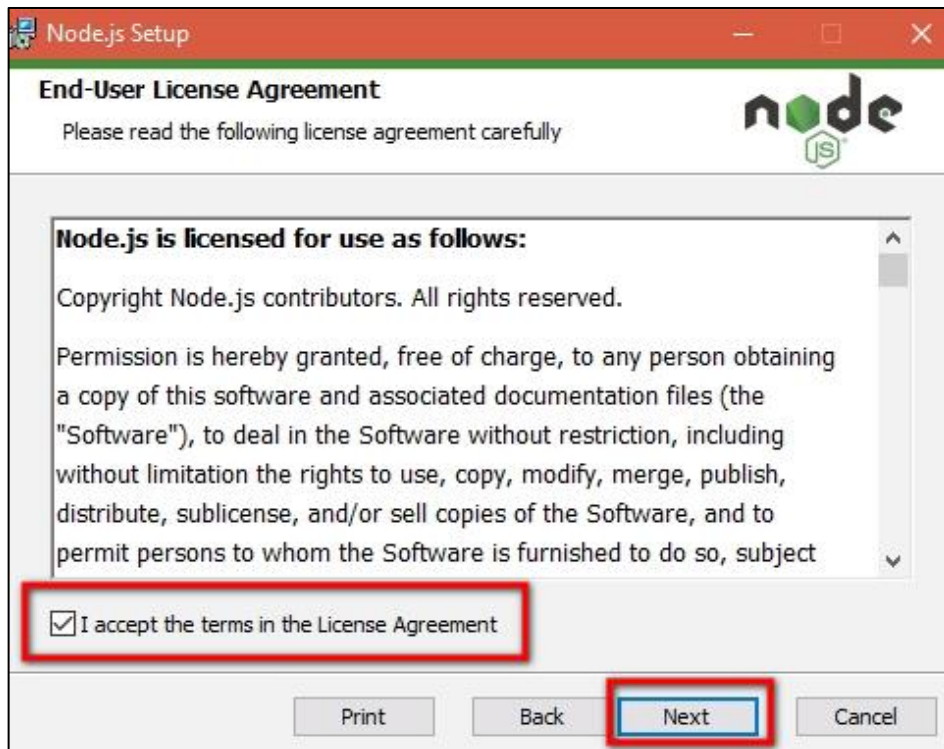


Source Code

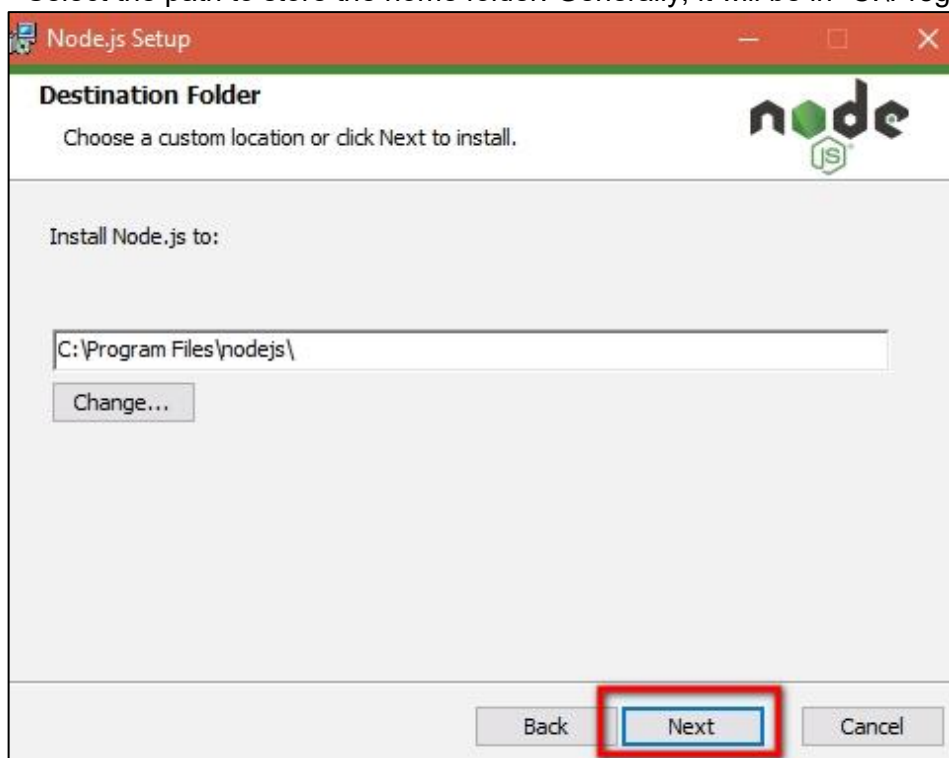
node-v8.9.3.tar.gz

Windows Installer (.msi)	32-bit	64-bit
Windows Binary (.zip)	32-bit	64-bit
macOS Installer (.pkg)	64-bit	
macOS Binaries (.tar.gz)	64-bit	
Linux Binaries (x86/x64)	32-bit	64-bit
Linux Binaries (ARM)	ARMv6	ARMv7
Source Code	node-v8.9.3.tar.gz	

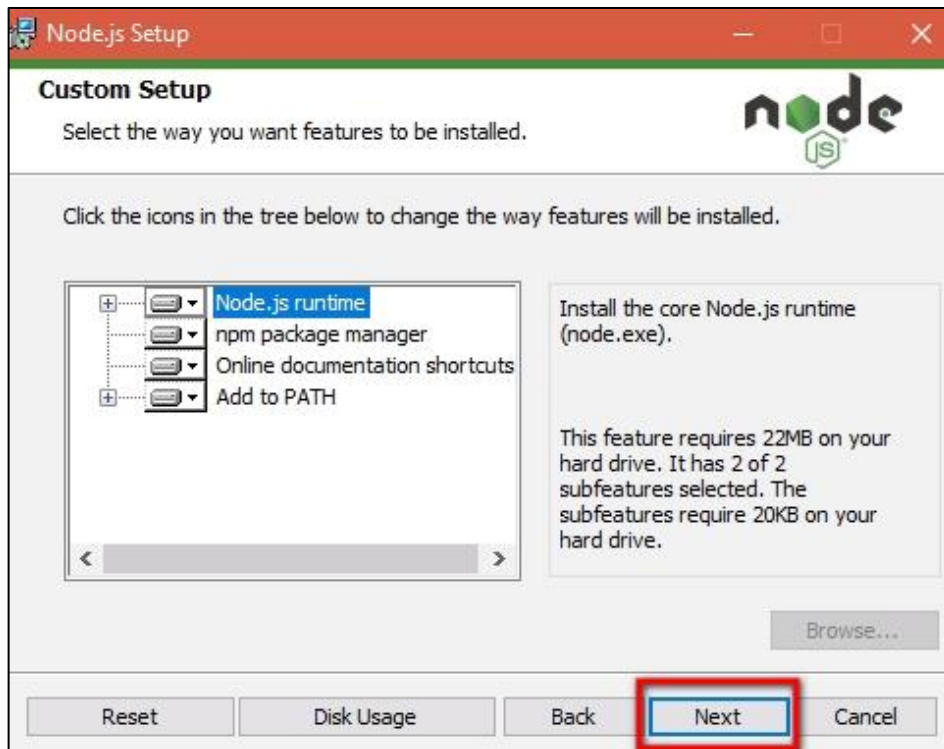
3. On clicking the link, it will automatically start downloading.
4. From the downloads click and run the installer.
5. Accept the license agreement and click next



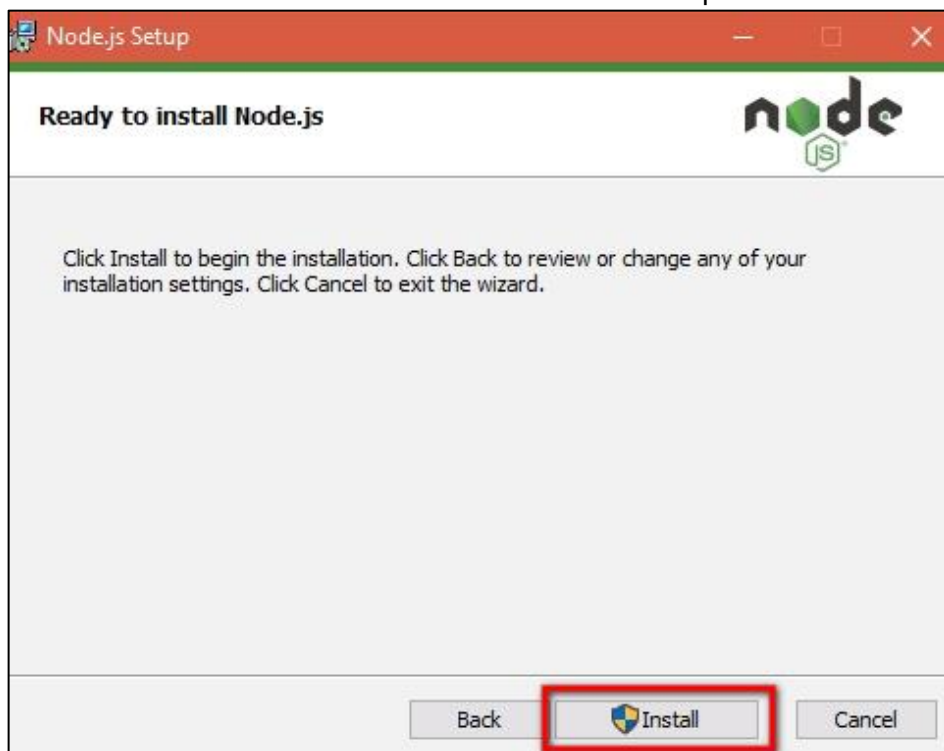
6. Select the path to store the home folder. Generally, it will be in "C:\Program Files\nodejs\".



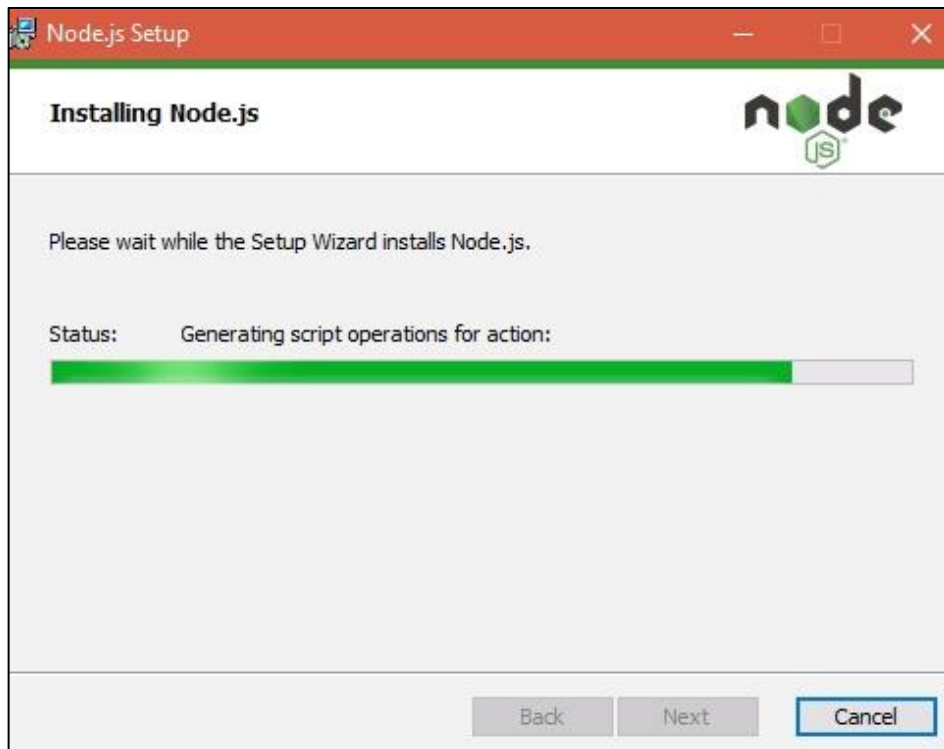
7. In the Custom Setup page, just click the next button.



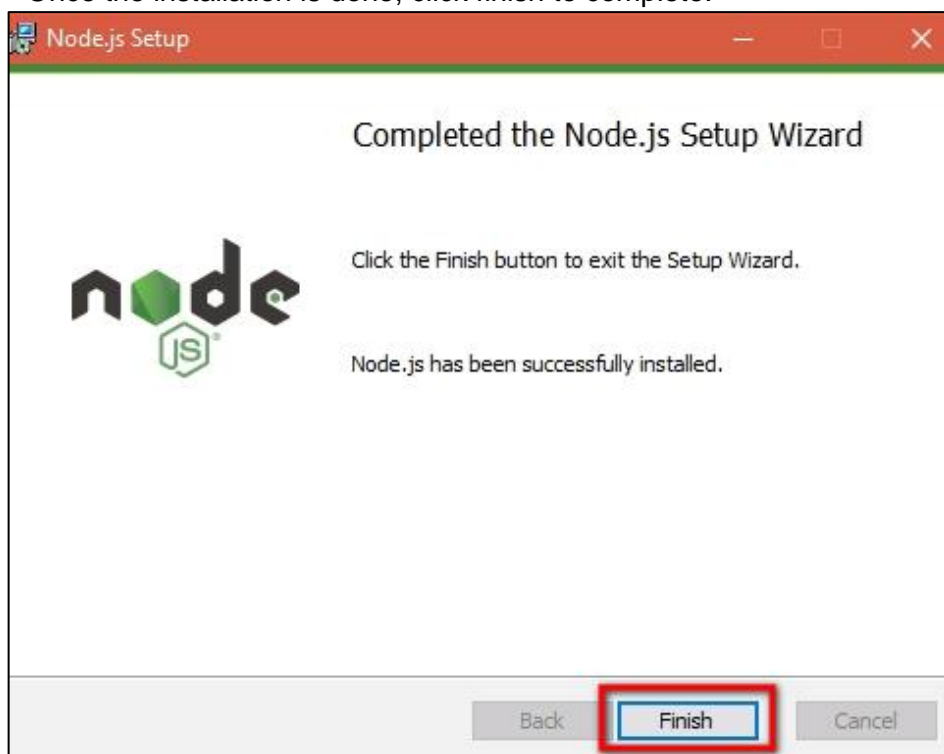
8. Just click the **"Install"** button to start the installation process.



9. Then it will start the installation.



10. Once the installation is done, click finish to complete.



11. Open the command prompt, and type "node --version" to check whether the node is installed correctly or not. If it gives the version then it is installed perfectly.


```
C:\Windows\System32\cmd.exe
Microsoft Windows [Version 10.0.16299.192]
(c) 2017 Microsoft Corporation. All rights reserved.

C:\Users\Admin>node --version
v8.9.3

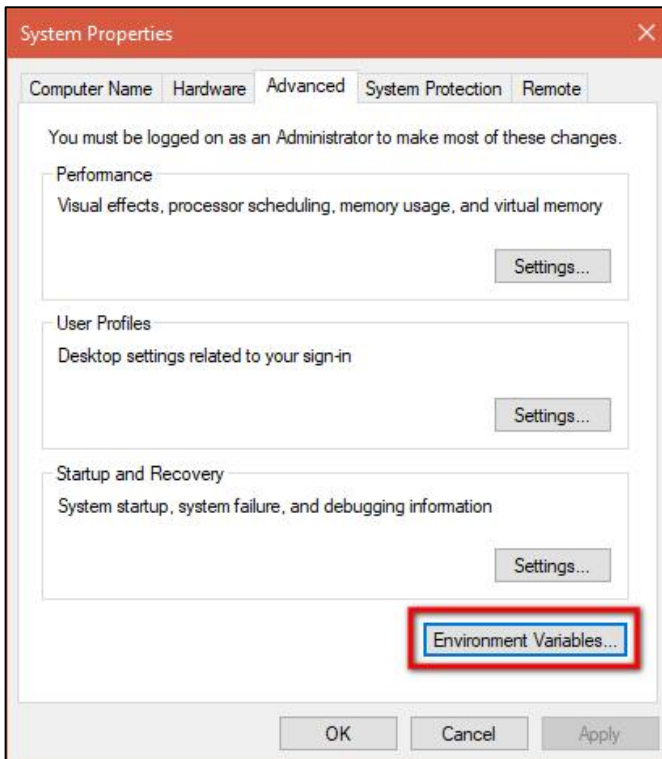
C:\Users\Admin>
```

7.1 SETTING ENVIRONMENTAL VARIABLE

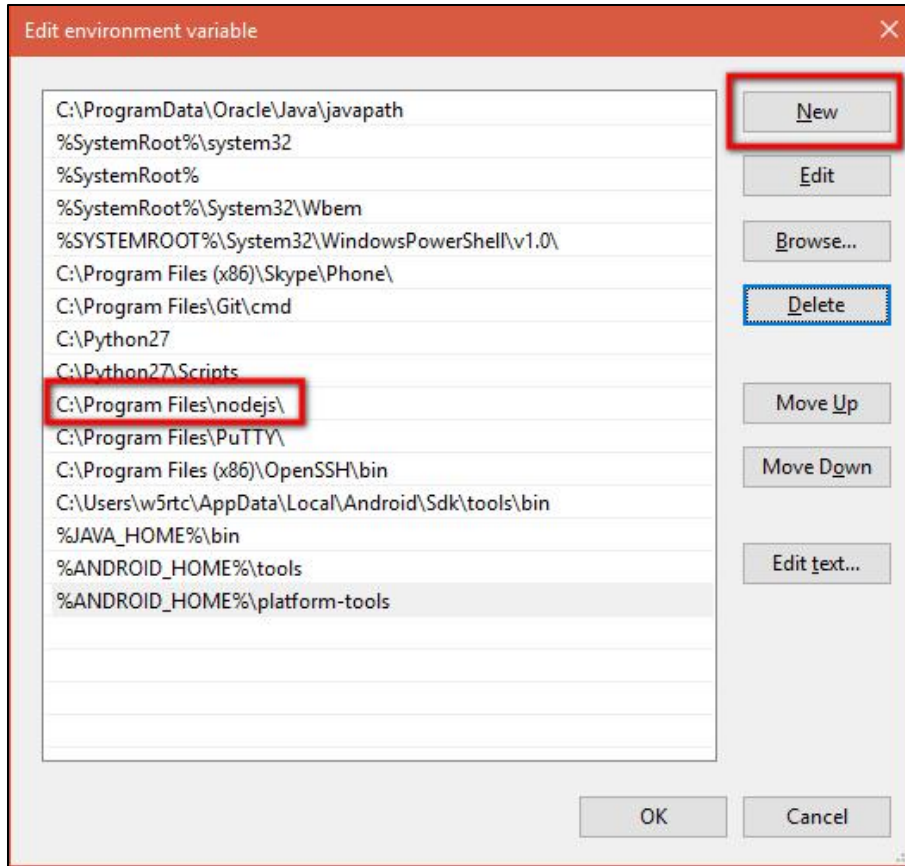
1. To set environmental variable and path, right click on My Computer > Properties > Advanced System Settings.



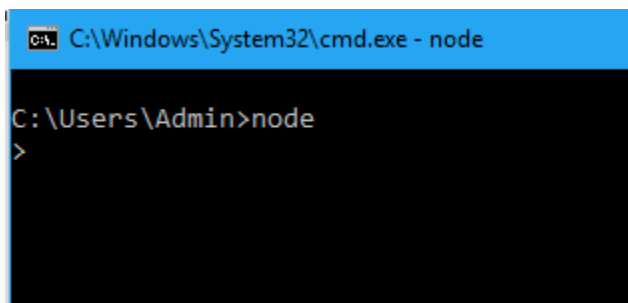
2. Click on the Environment Variables.



3. In the system variables section, click path and click new and enter the path as C:\Program Files\nodejs\



4. In order to check, open command prompt and type node and press enter. It should the node environment. Then press the Ctrl + C to exit.

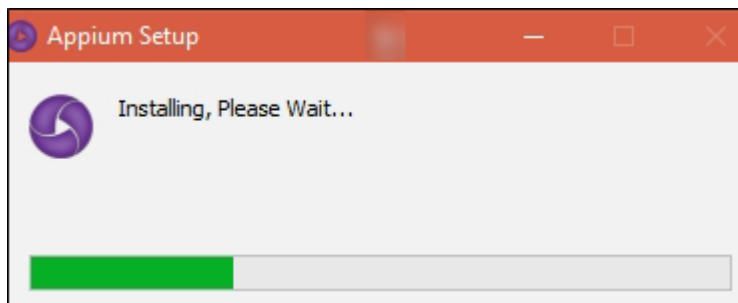


8. INSTALLATION – APPIUM

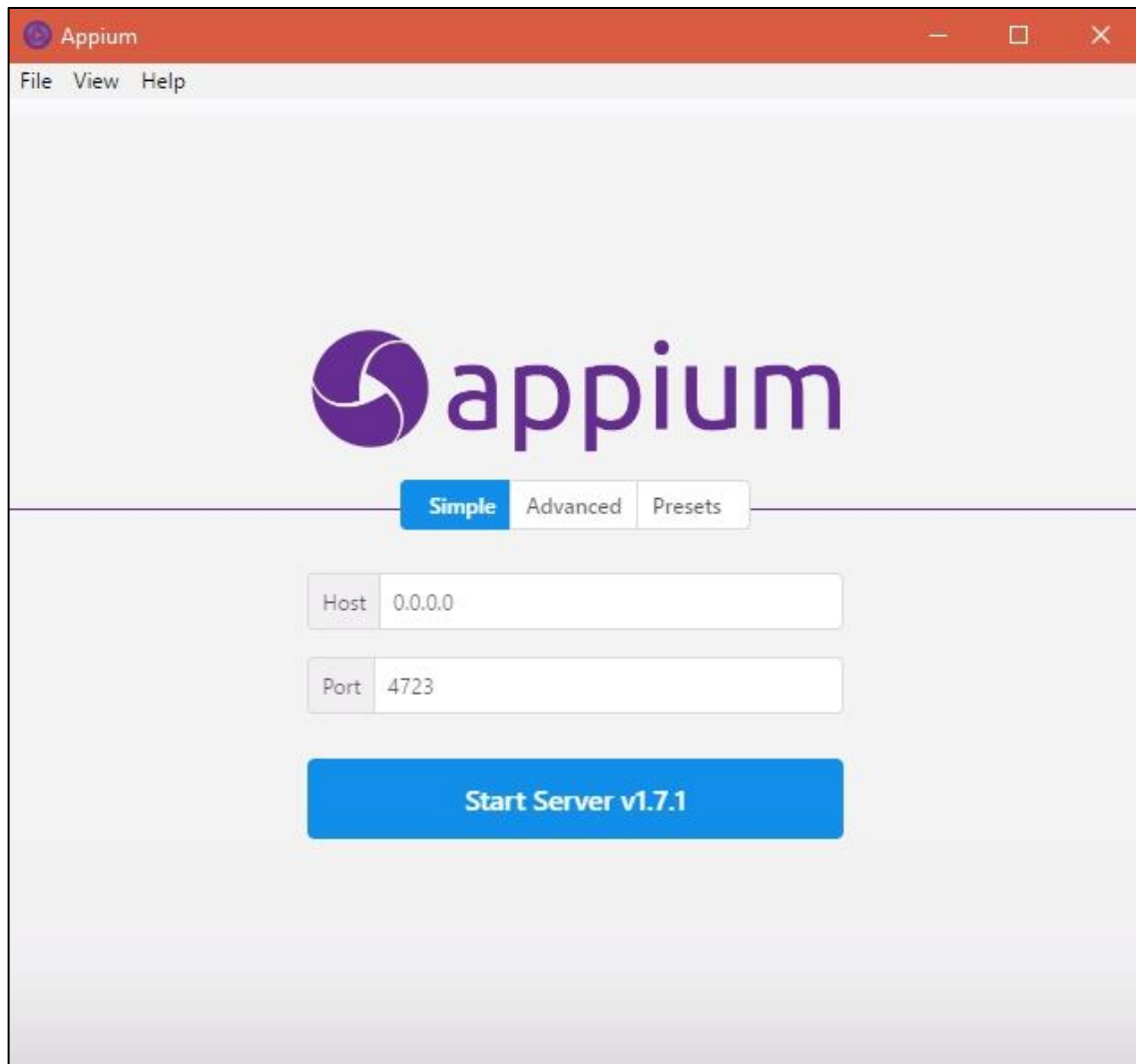
1. Click the link to download:
<http://appium.io/>



2. Run the installer.



3. Once it is installed, it will open the Appium.



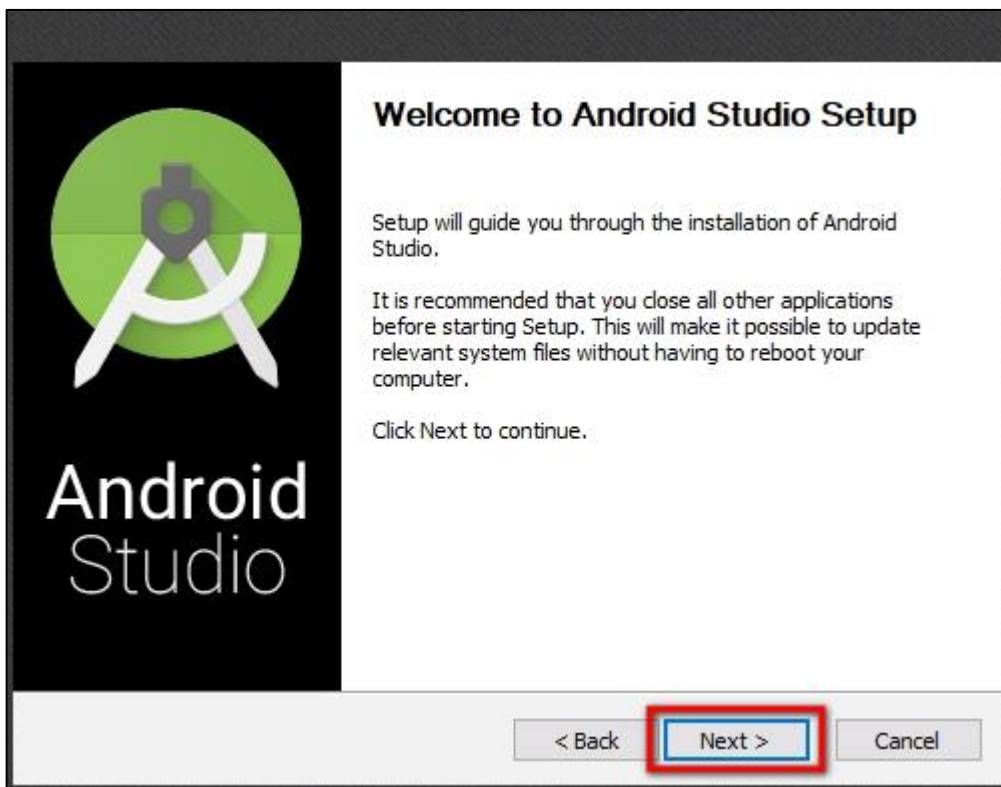
4. Enter the host and port number and start the server.

9. INSTALLATION – ANDROID STUDIO

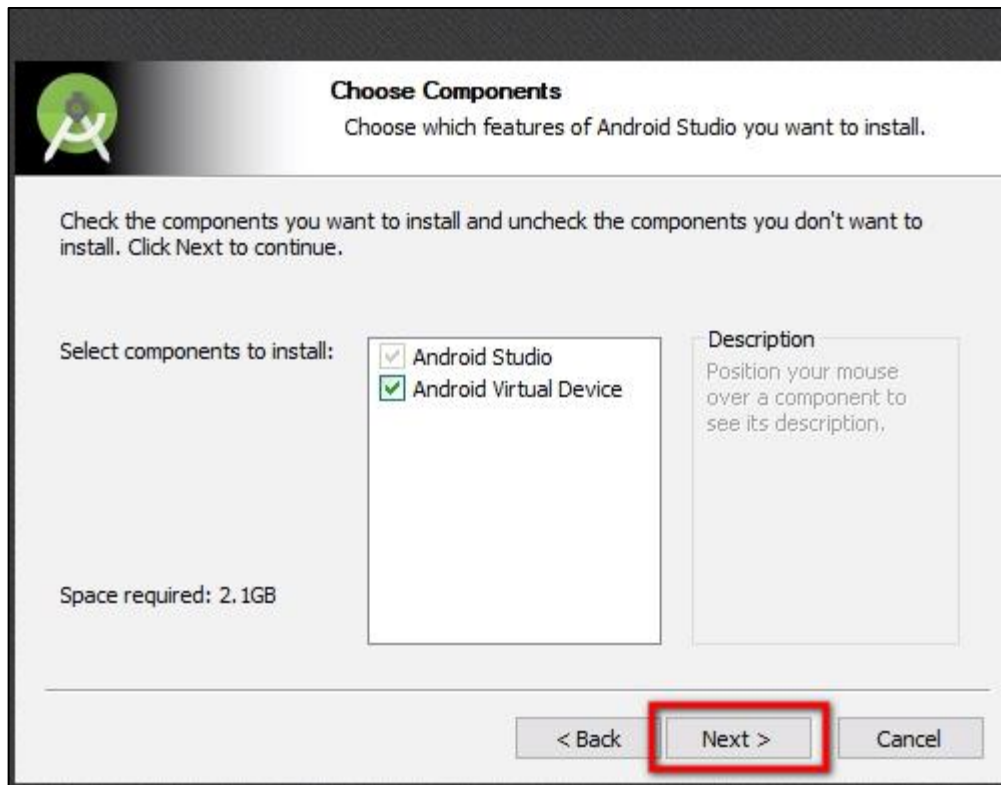
1. Download the android studio installer from the link below:
<https://developer.android.com/studio/index.html>
2. Run the downloaded installer.



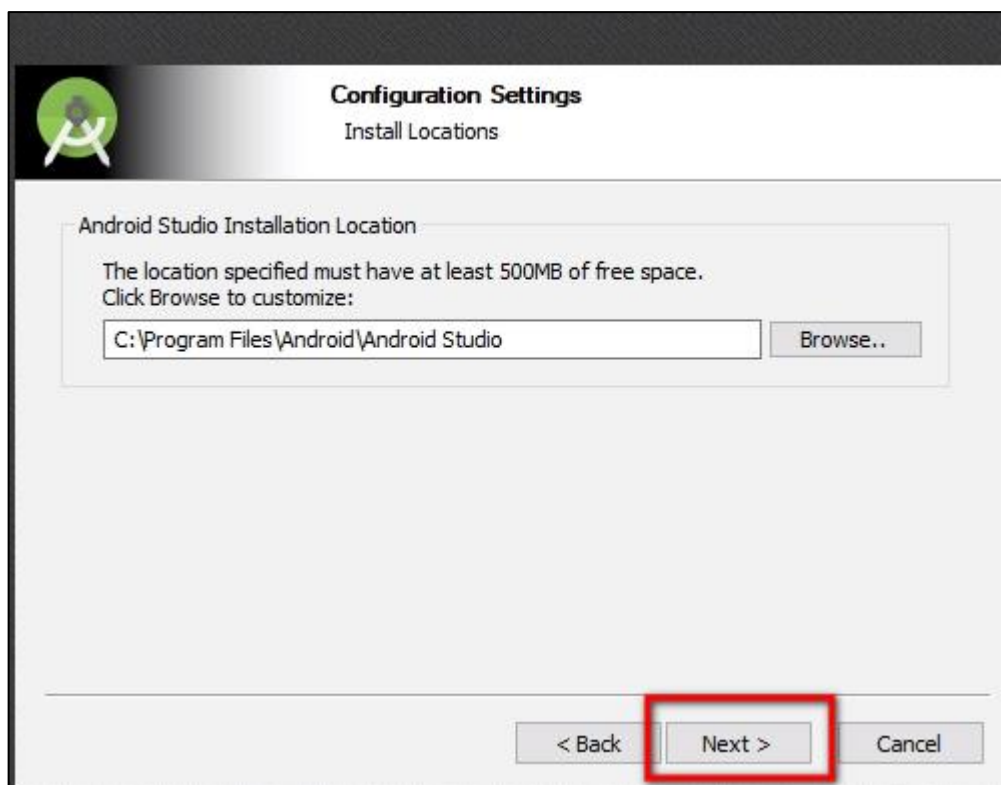
3. Click Next in the welcome page of the android studio setup.



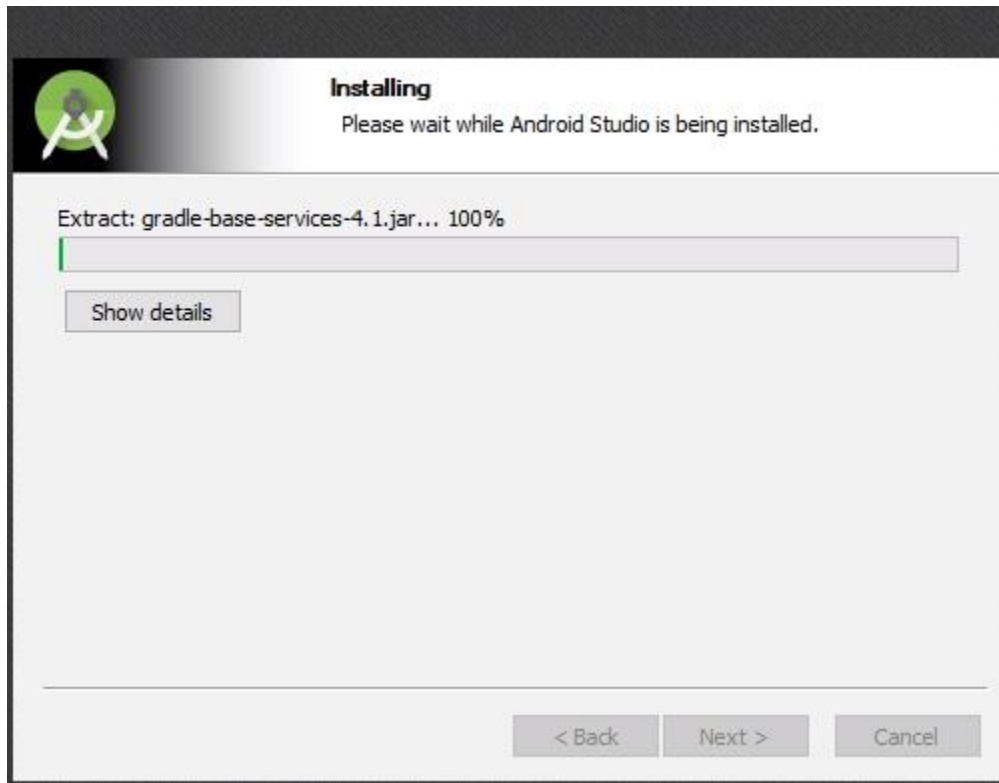
4. Select the components that are to be installed i.e. the android studio and android virtual device.



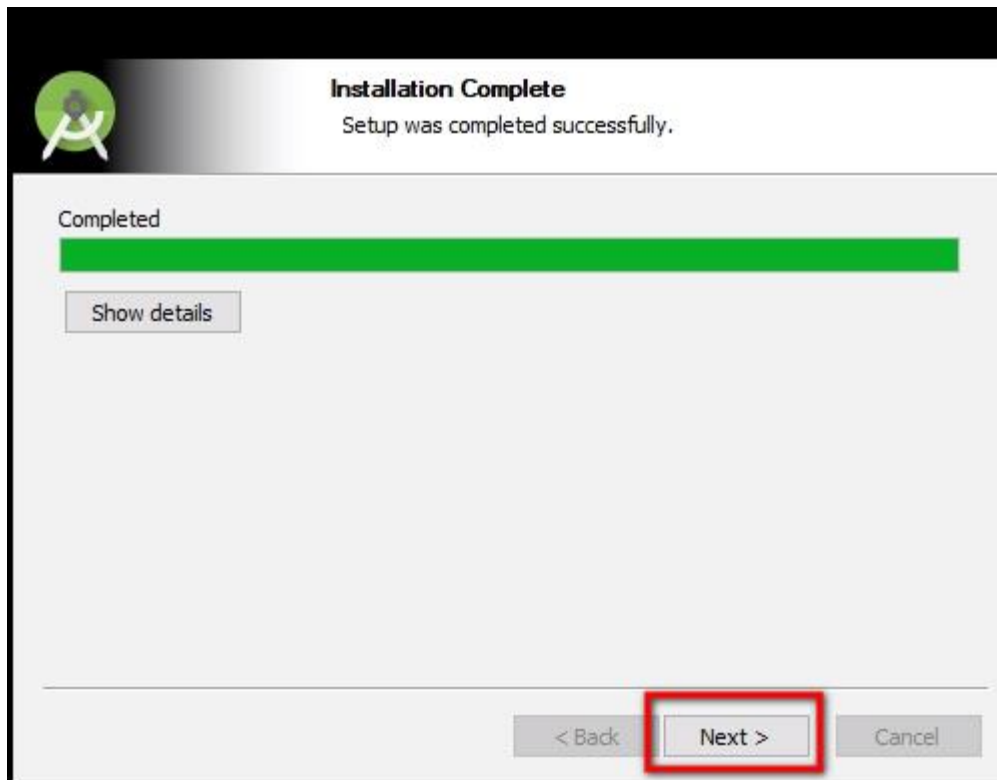
- Click Next to select the path of the android studio.



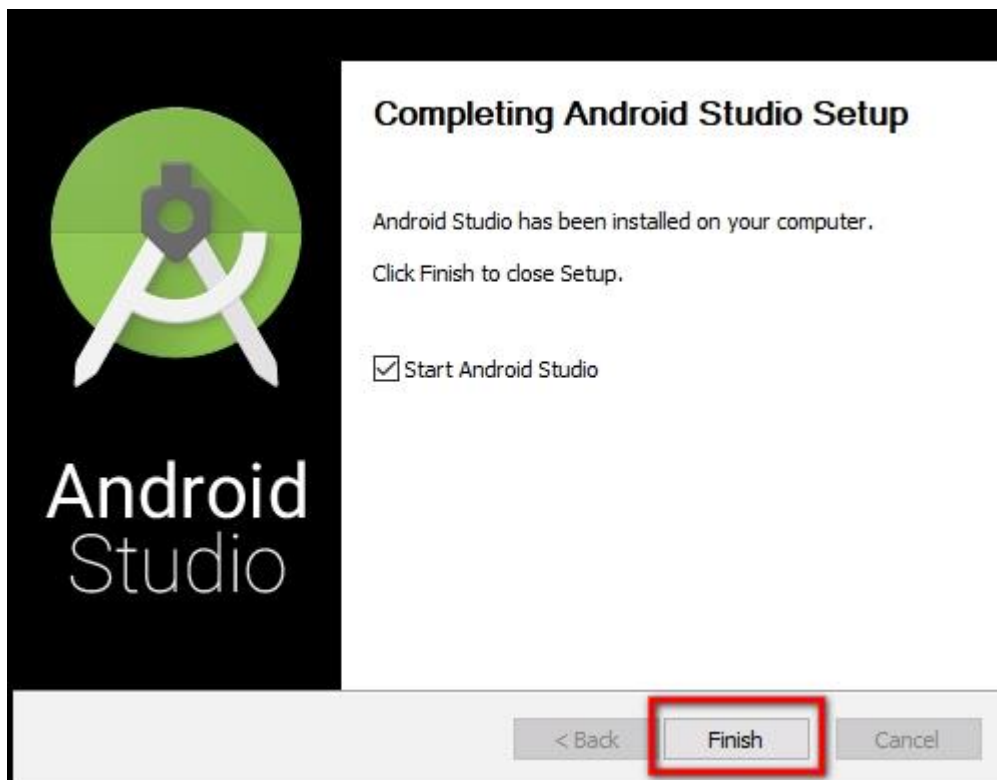
6. On clicking the Next button, it will start installing.



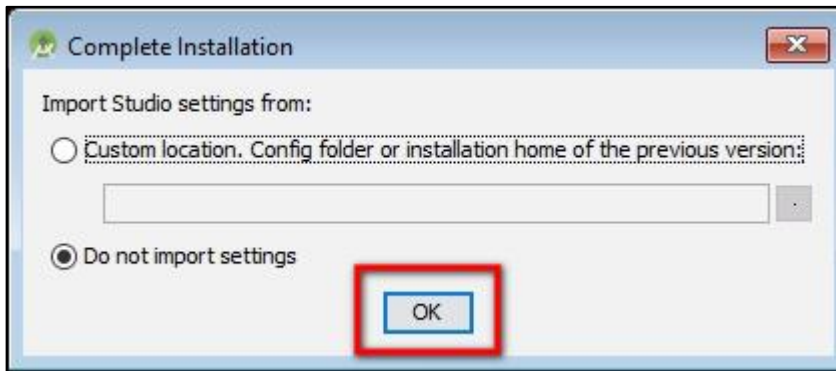
7. Once the installation is complete, click Next.



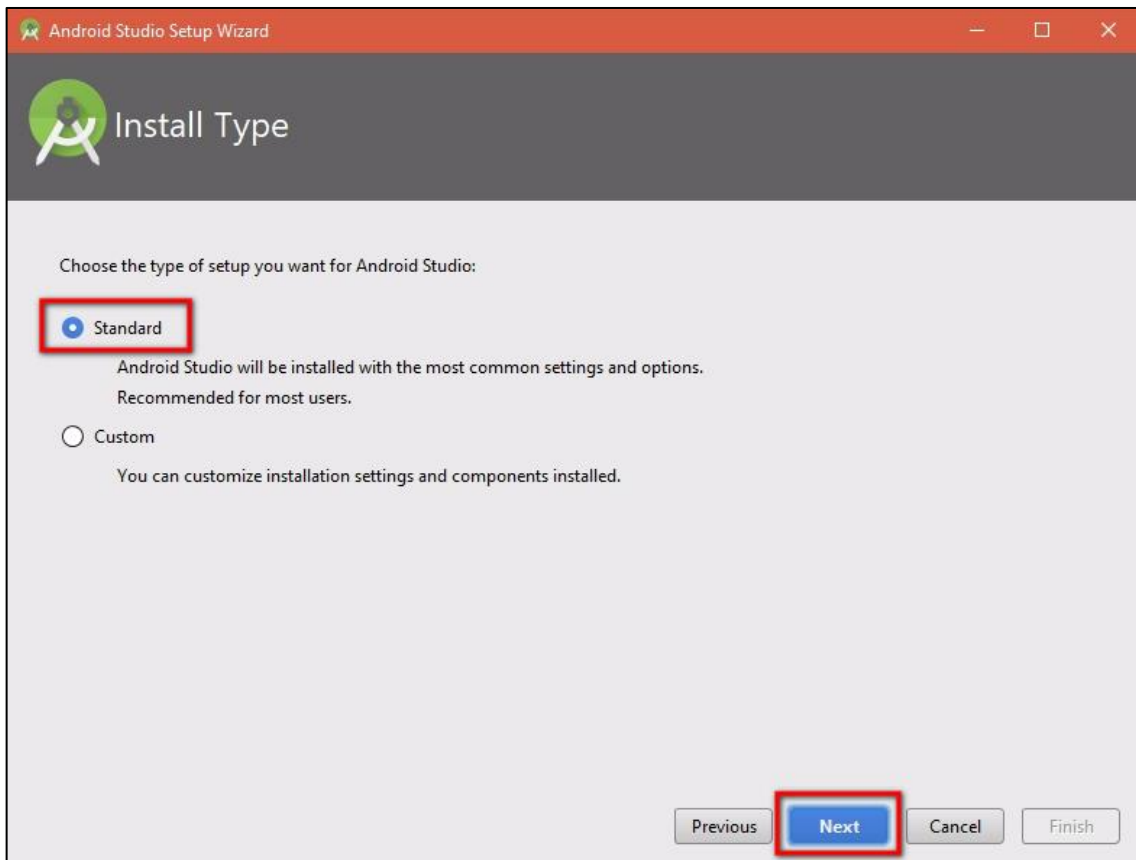
8. Clicking the finish button will finish the setup and open the android setup.



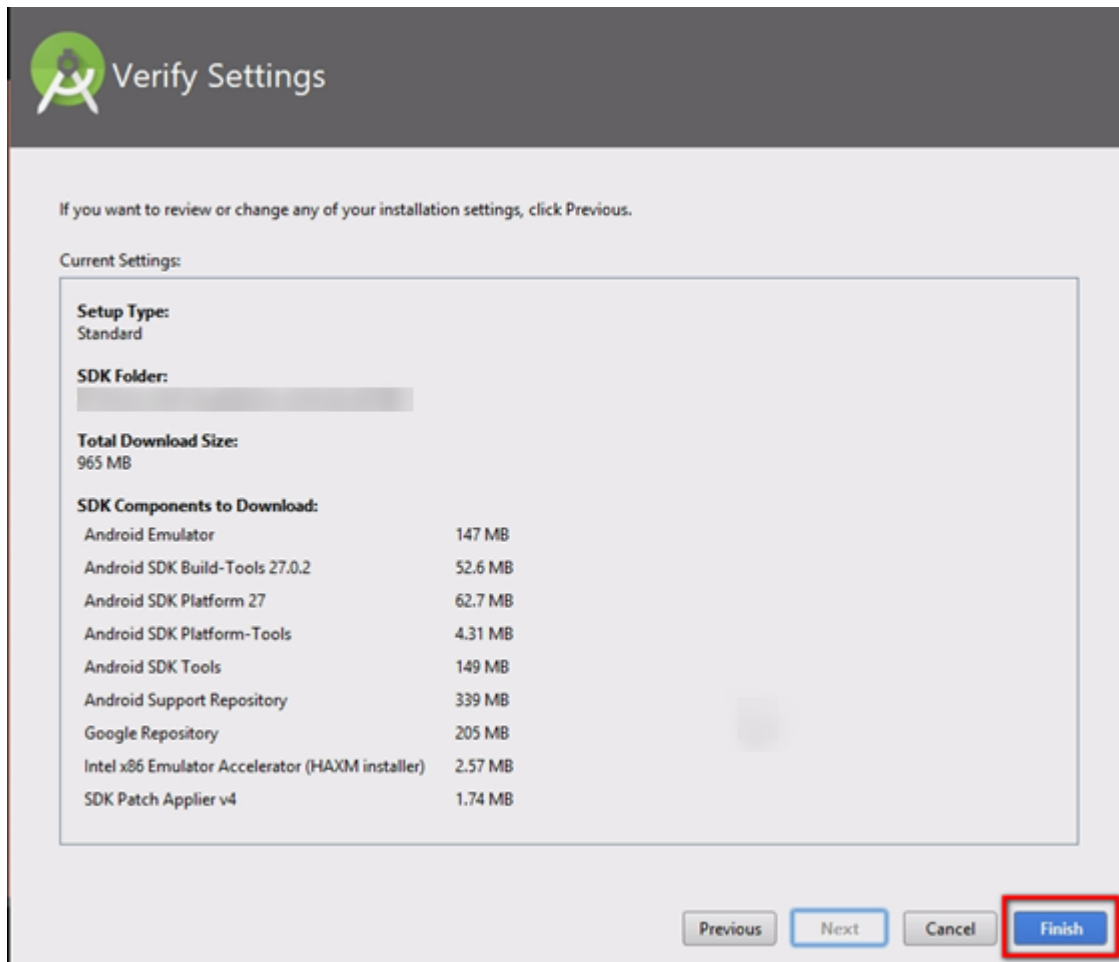
9. It will ask whether the user want to import the setting or not.



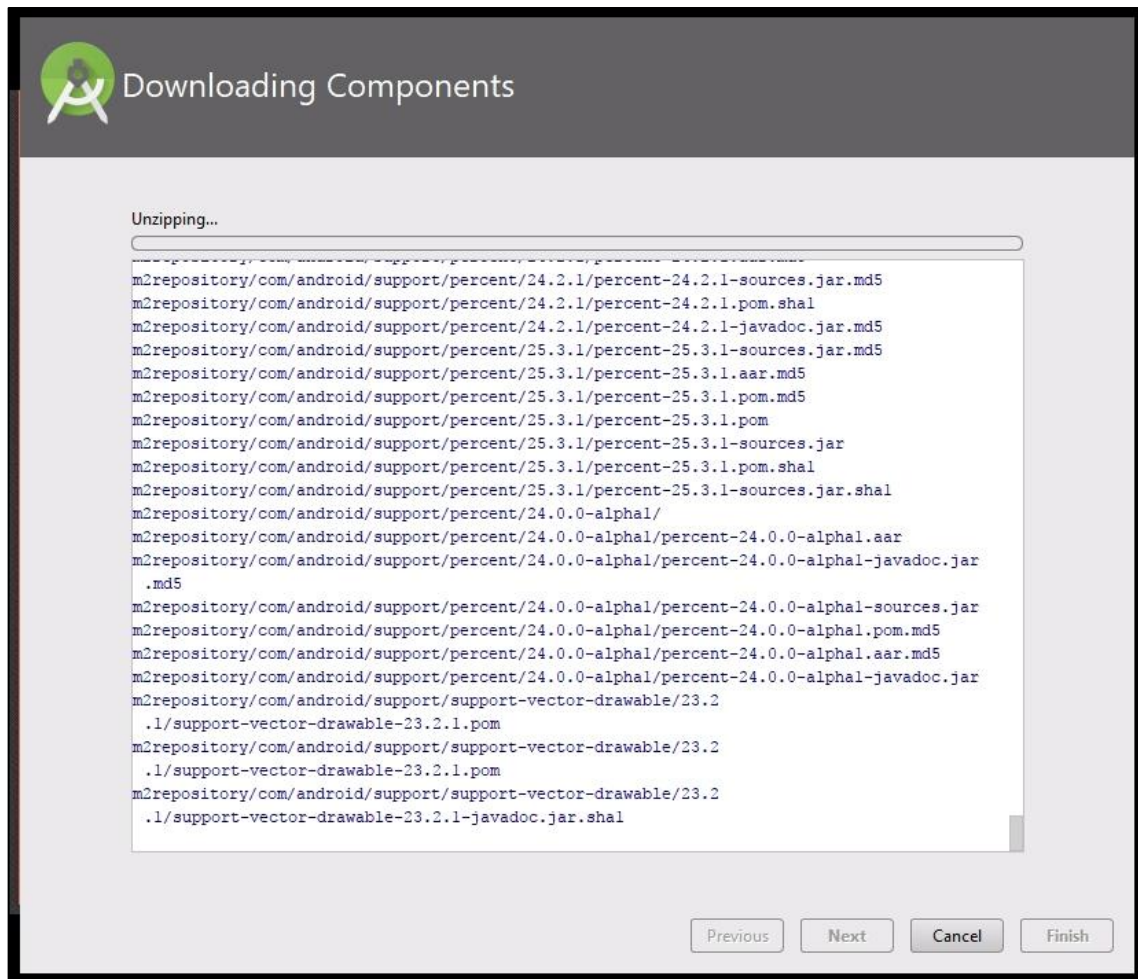
10. Select the standard option to install.



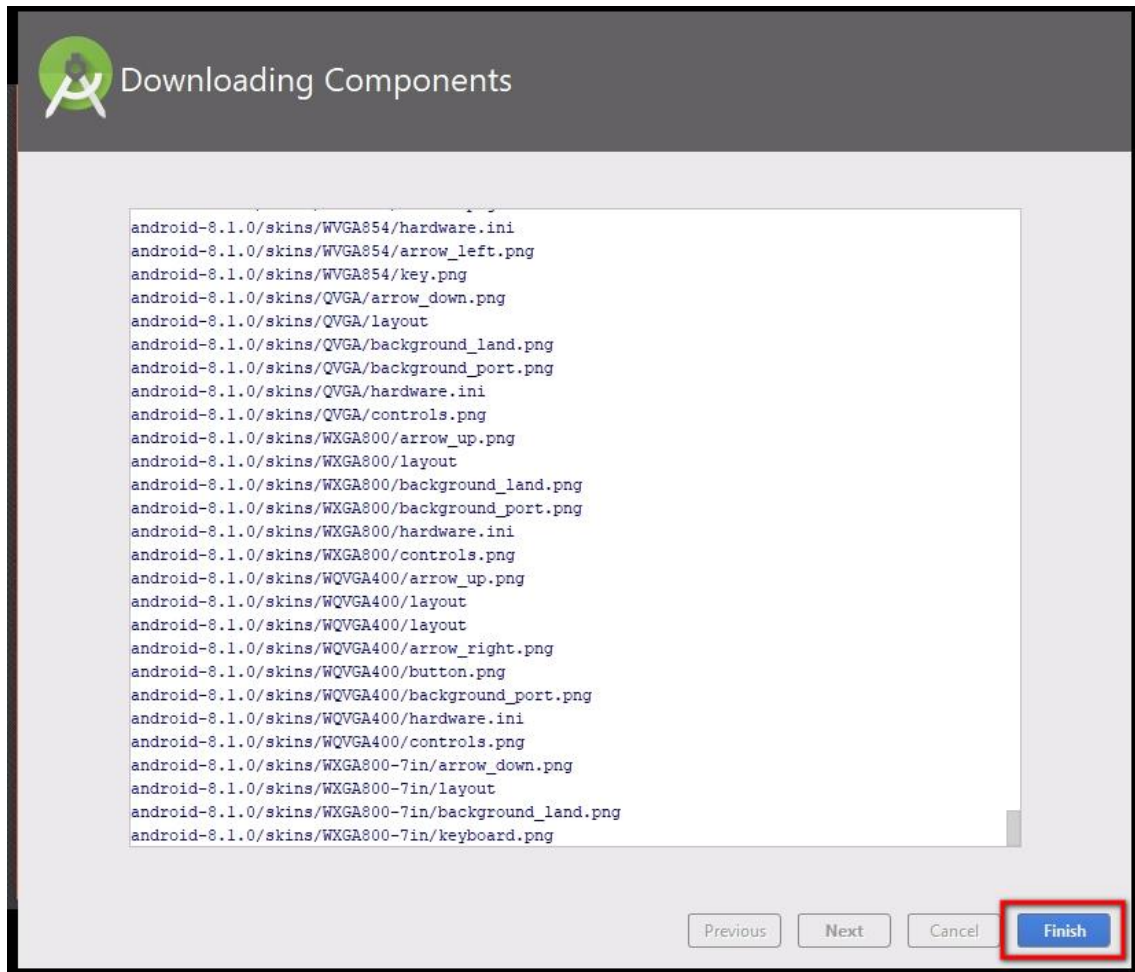
11. It will ask to install the SDK packages, install by clicking the Finish button.



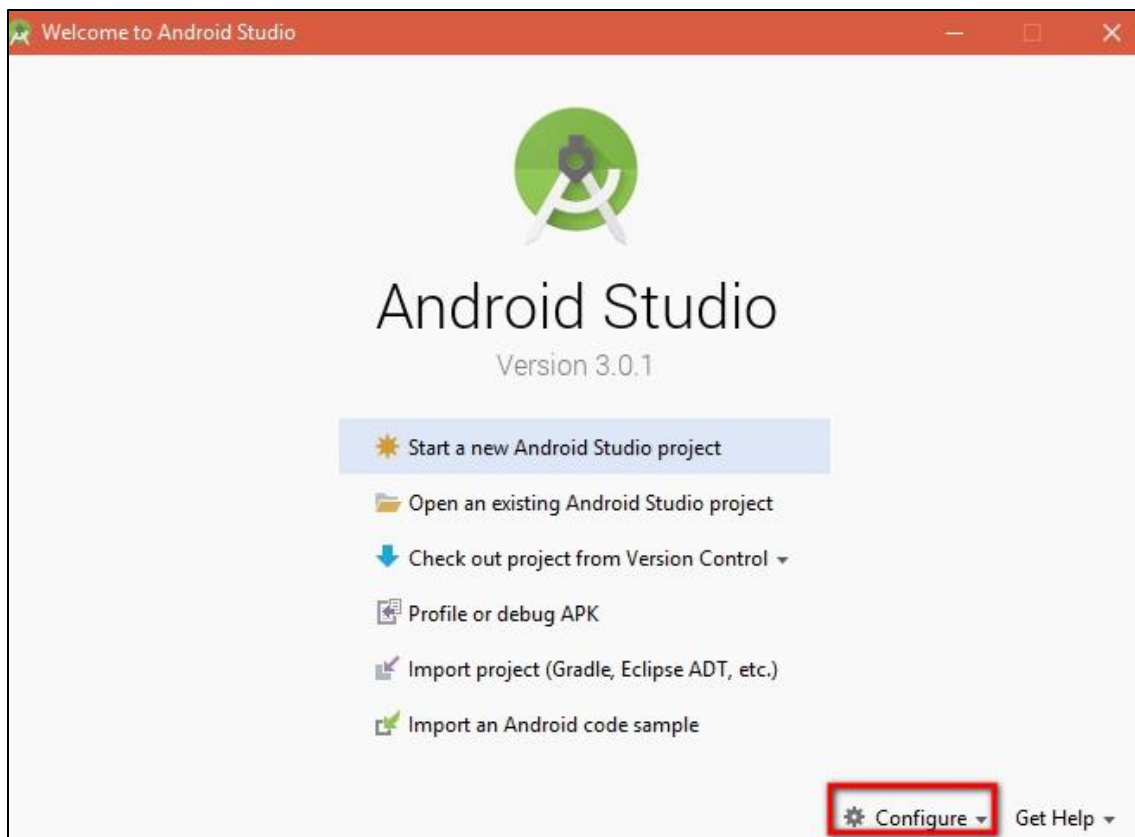
12. It will start to download.



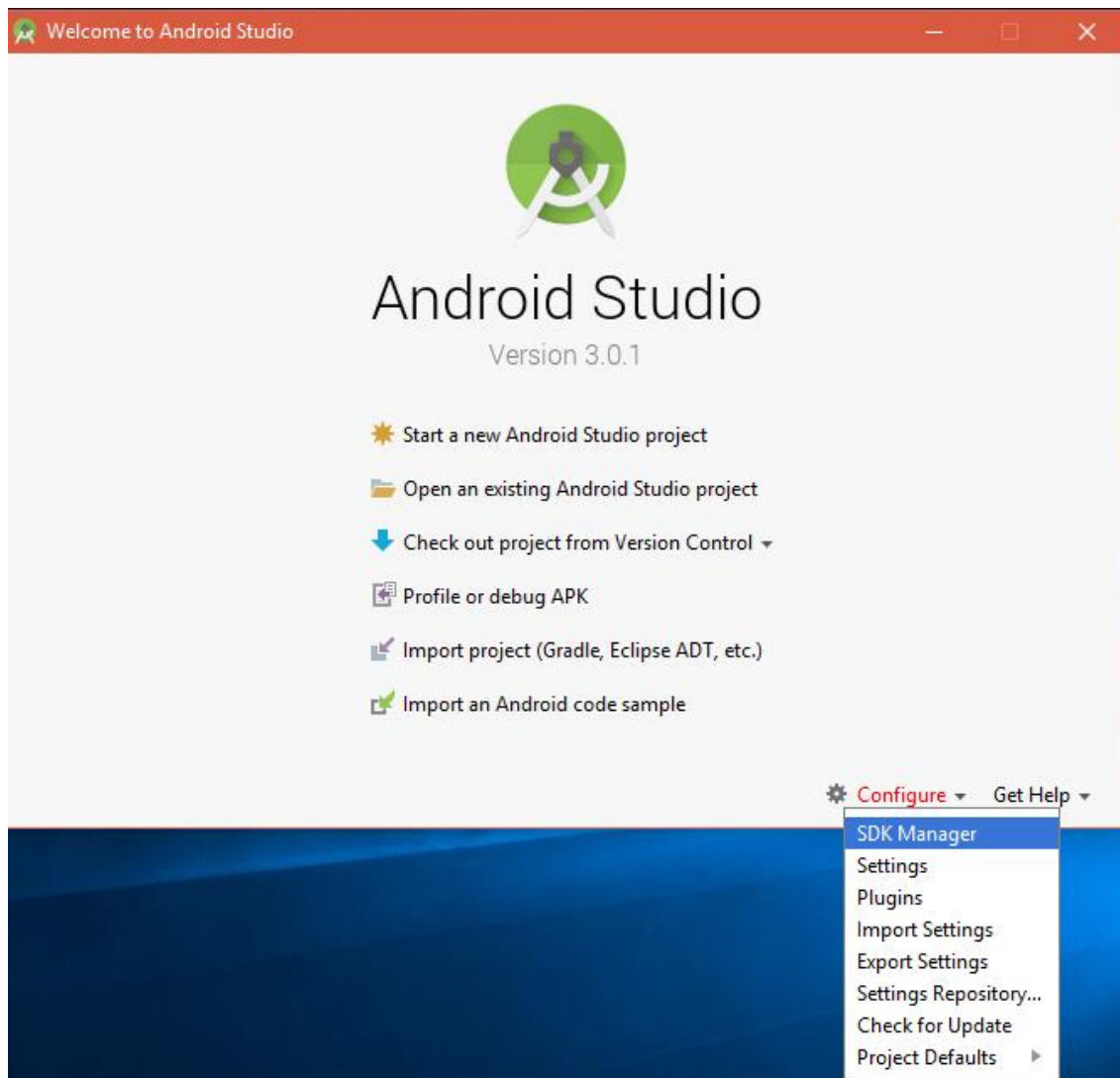
13. Once download is finished, click Finish.



14. Once finish is clicked, it will open the android studio.

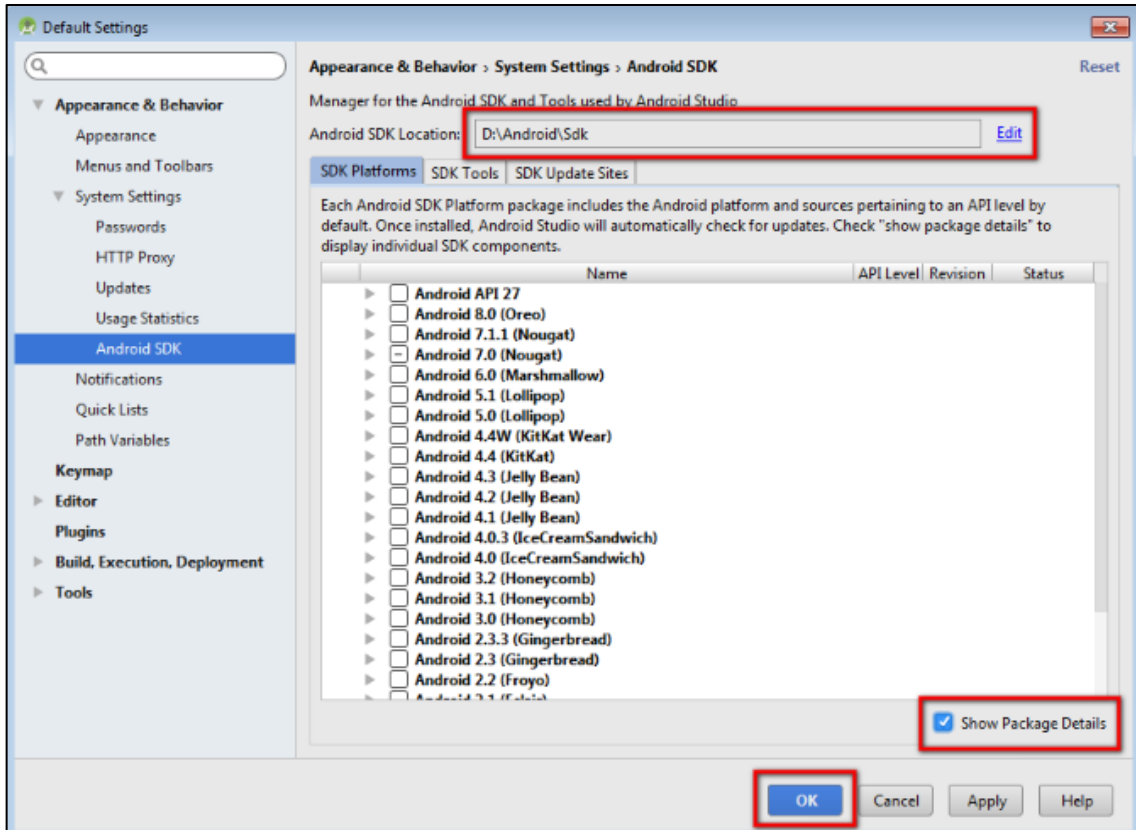


15. Under the configure dropdown, click the SDK Manager.

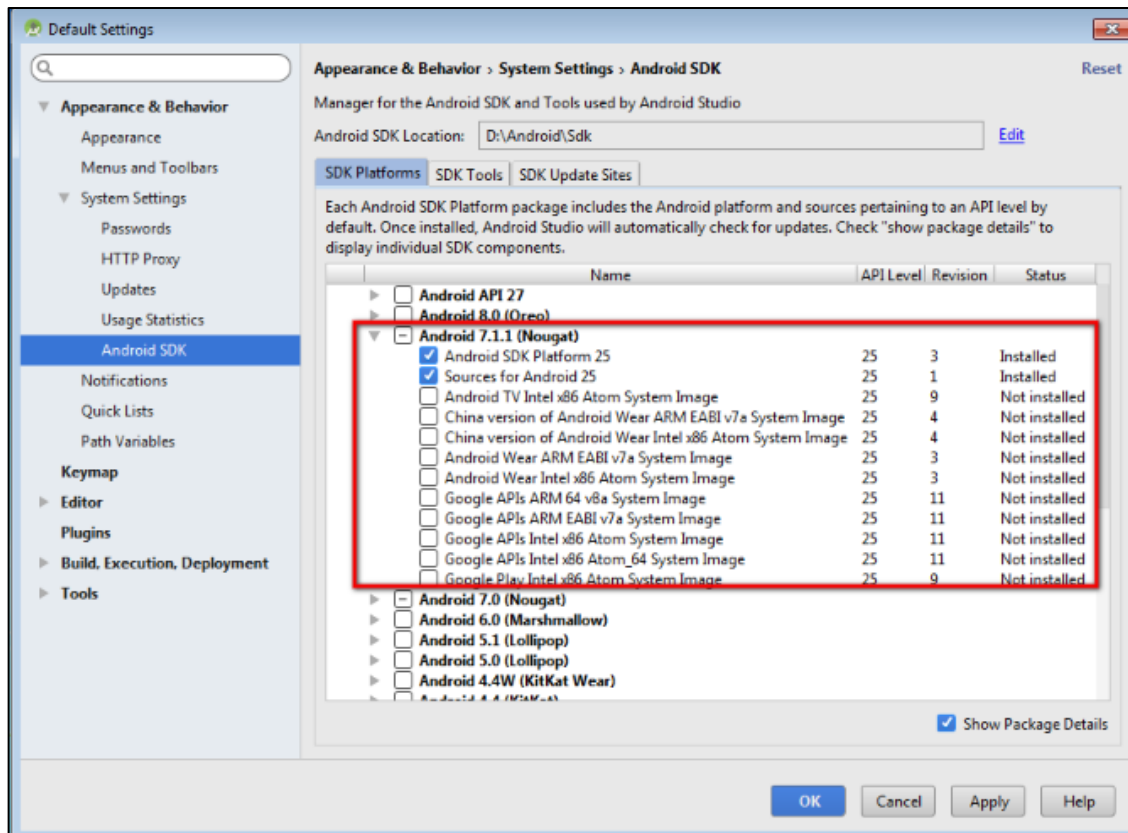


16. On clicking the SDK Manager, a dialog box opens up and the user is allowed to select the SDK's.
17. Click the Show Package Details to see all the packages.
18. In the top of the dialog, the file location of the SDK packages is given. The user can change the location based on the storage space available. The example shows D: drive.

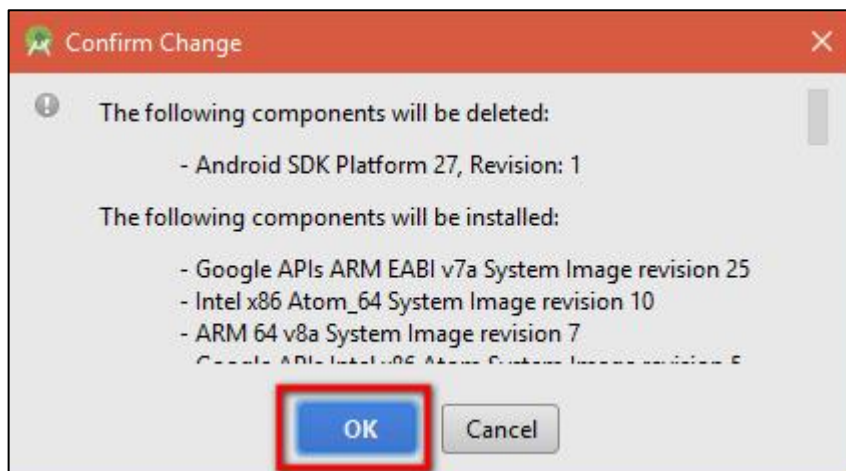
The user has to change folder location to some other drive in which there is more space. The user must not install in C: drive.



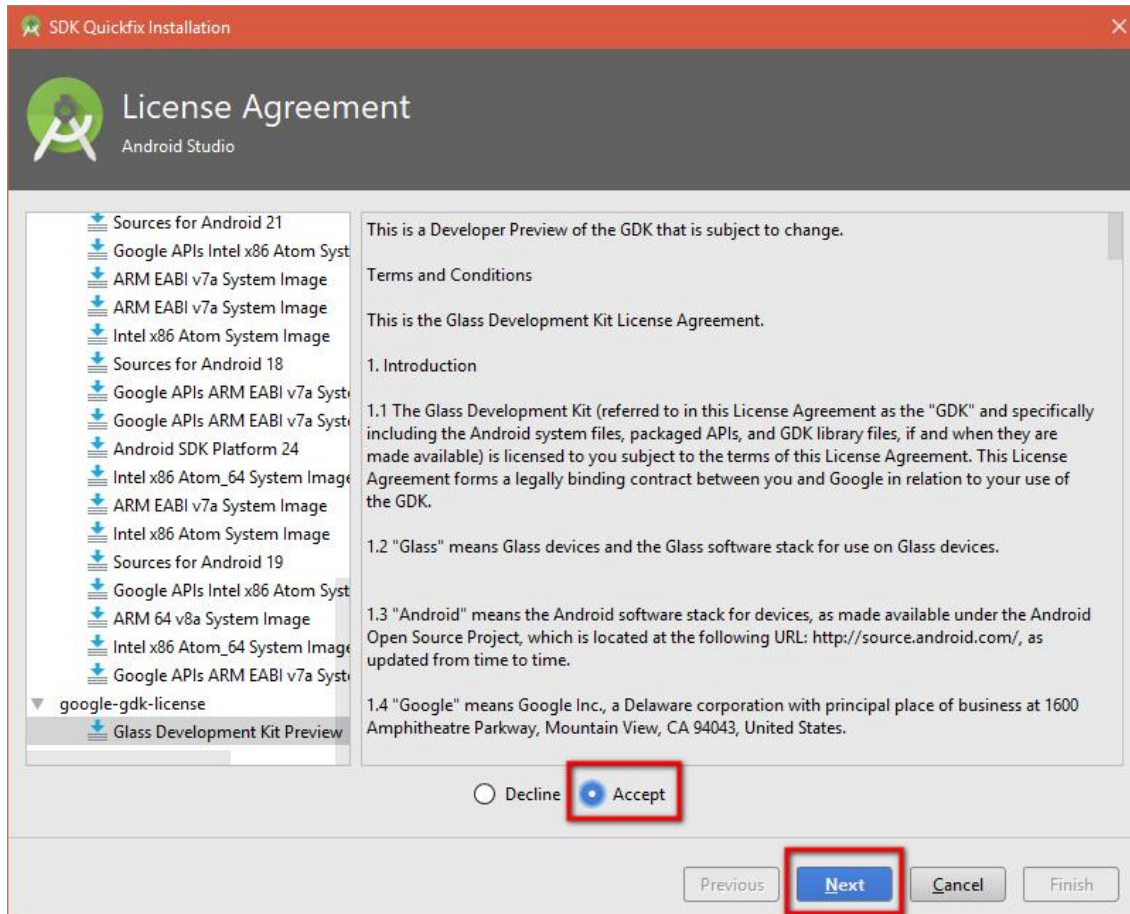
19. In the SDK's list, select the following SDK packages:
 - a. Android 7.1 (Nougat)
 - b. Android 7.0 (Nougat)
20. For the above SDK's, select all the packages under each android version except the TV and Wear packages.



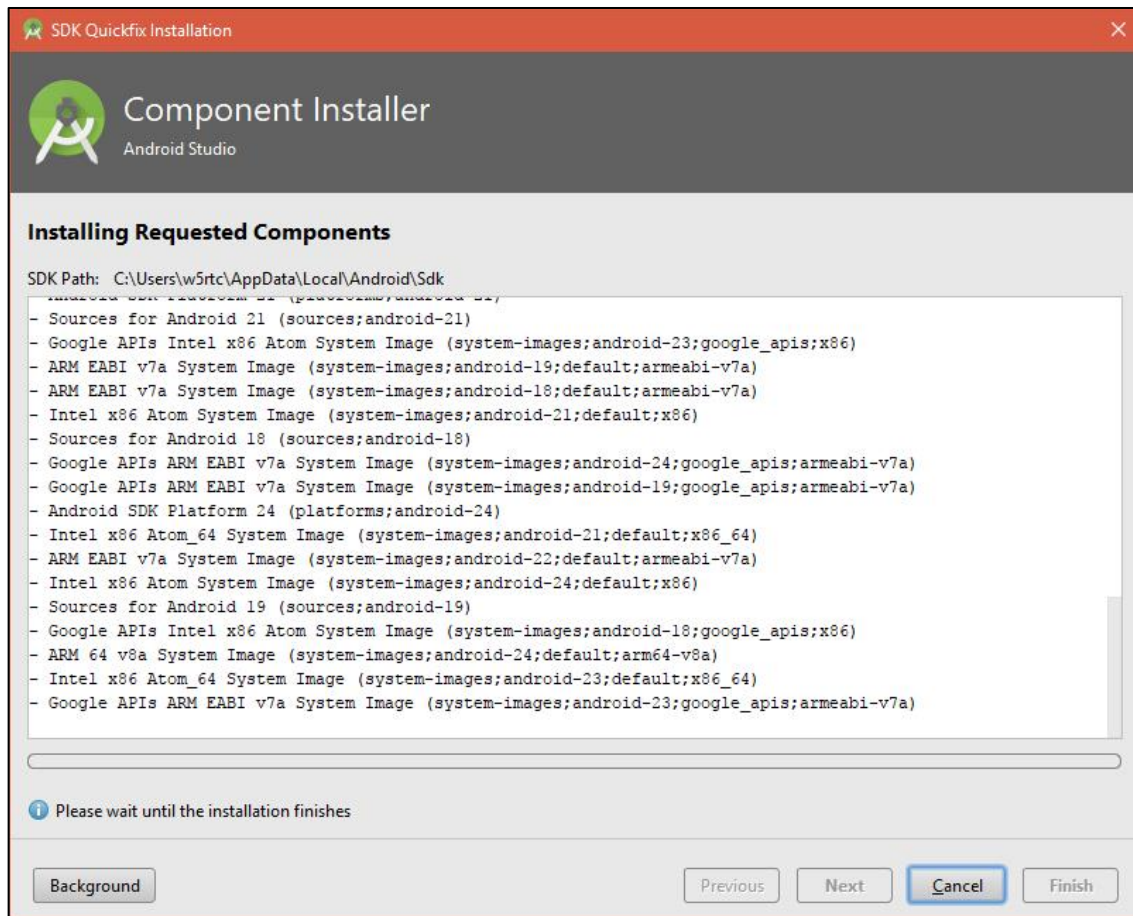
21. Once all the packages are selected, click Ok button.
22. Other packages of android versions are also available, the user can install based on the requirements.
23. It asks for the confirmation, click OK button.



24. Once confirmation is given to download, accept the license agreement and click the Next button to start to download.



25. It will start to download. It will take some time to download.



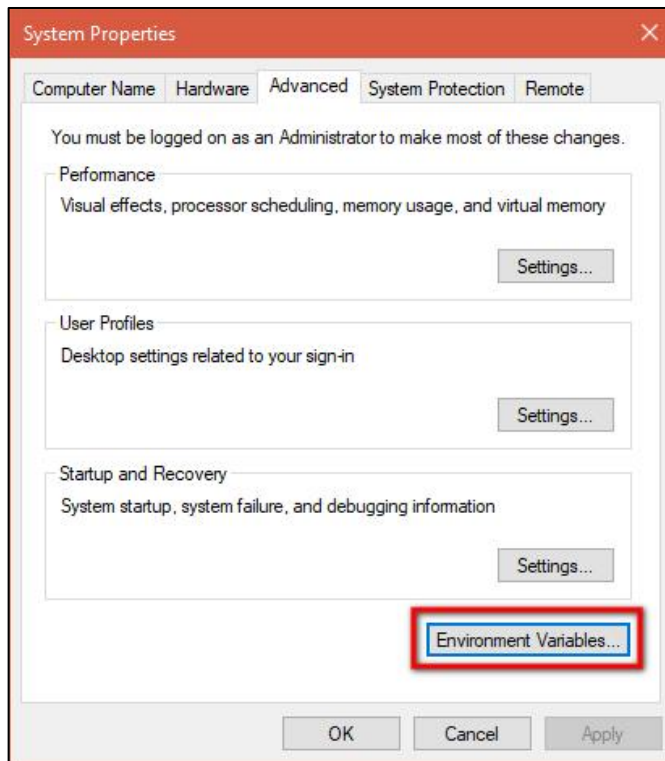
26. Once download is done, click Finish button.

9.1 SETTING UP ENVIRONMENTAL VARIABLES

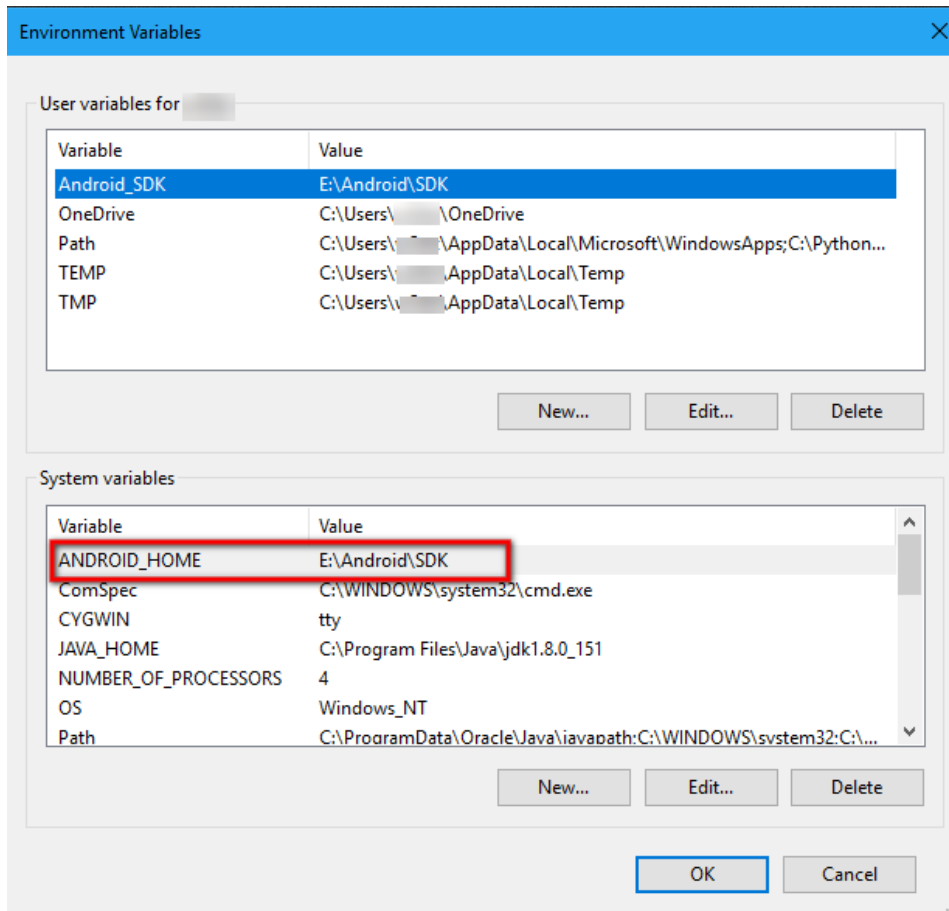
- a. To set environmental variable and path, right click on My Computer > Properties > Advanced System Settings



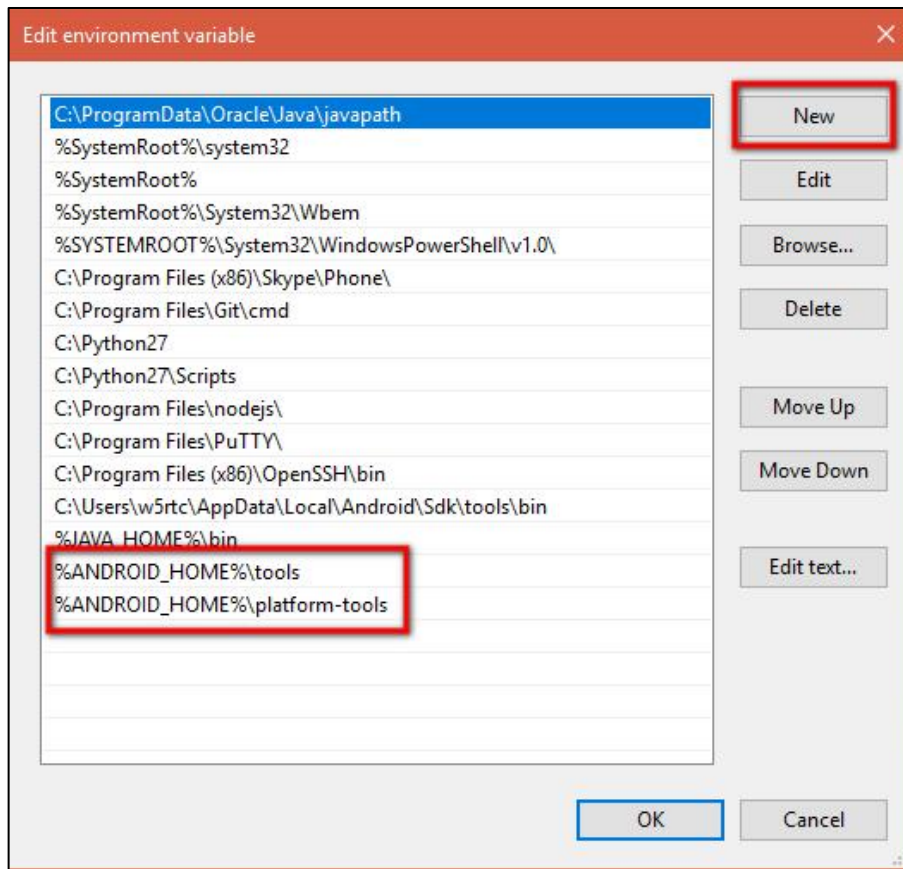
- b. Click on the Environment Variables



- c. In the system variables section, click new button and enter the following:
 - i. For variable name give ANDROID_HOME
 - ii. For variable value give the location of the SDK folder



- d. In the system variables section, click path and click new and enter the following:
- %ANDROID_HOME%\tools
 - %ANDROID_HOME%\platform_tools



- e. To check whether the android is working properly, open command prompt and type android.

```
C:\Users\Admin>android
*****
The "android" command is deprecated.
For manual SDK, AVD, and project management, please use Android Studio.
For command-line tools, use tools\bin\sdkmanager.bat
and tools\bin\avdmanager.bat
*****

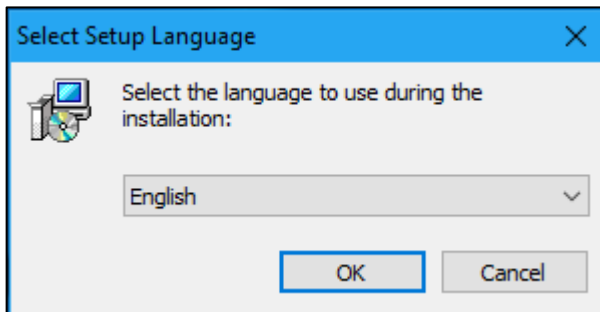
Invalid or unsupported command ""

Supported commands are:
android list target
android list avd
android list device
android create avd
android move avd
android delete avd
android list sdk
android update sdk
```

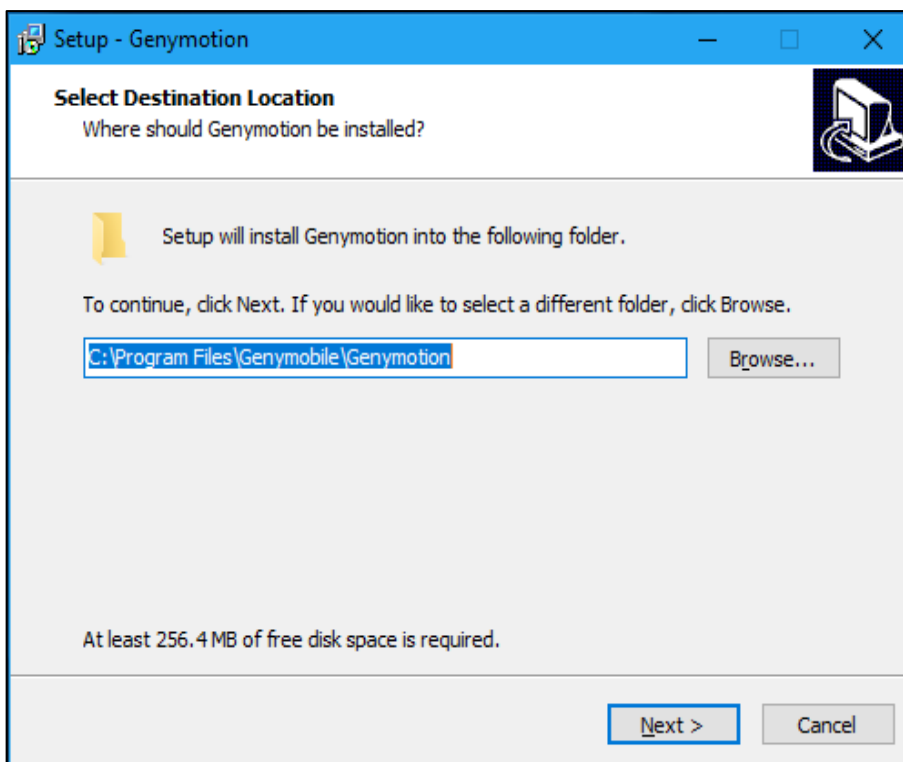

10. INSTALLATION – GENYMOTION

NOTE: Genymotion will not work without virtual box.

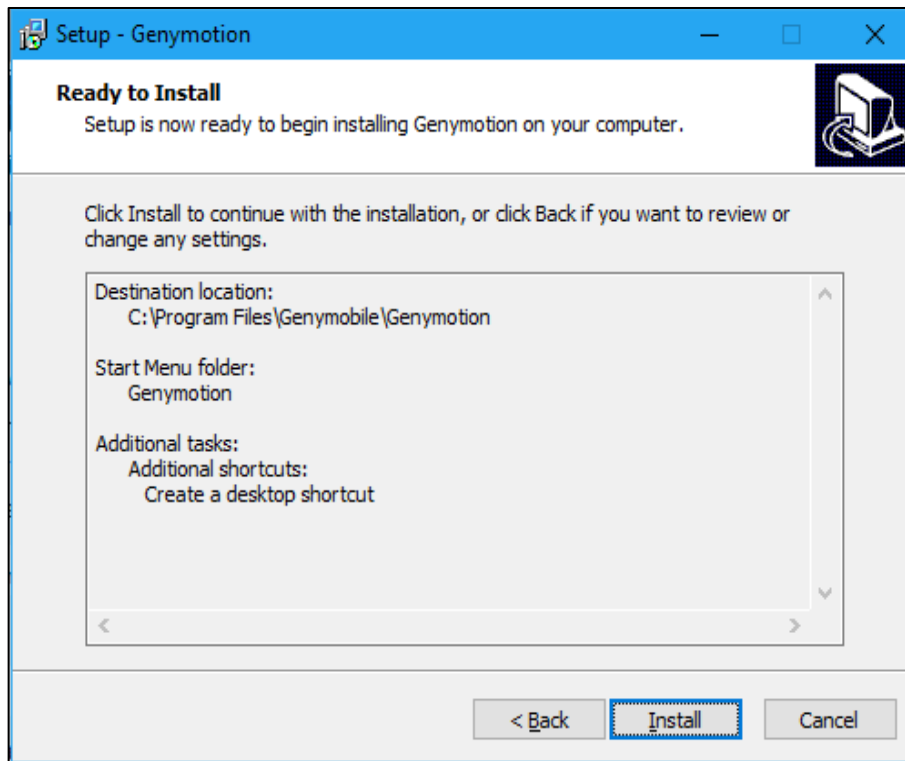
1. Click the link to download to Genymotion:
<https://www.genymotion.com>
2. The user has to signup first in order to download the genymotion.
3. Run the installer.
4. Select the language.



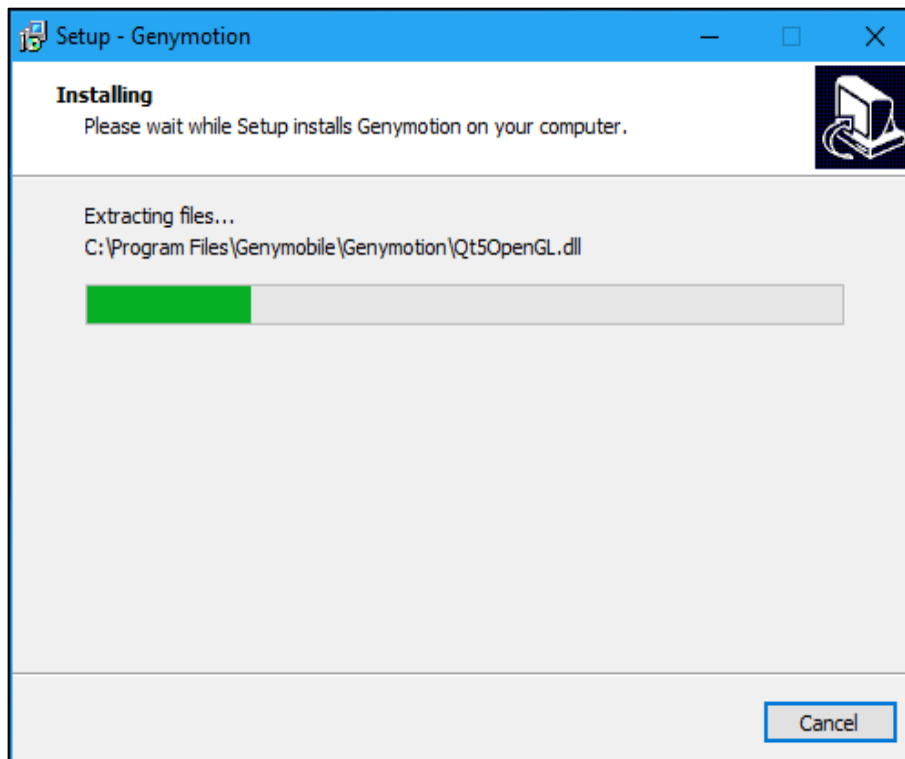
5. Set the folder location.



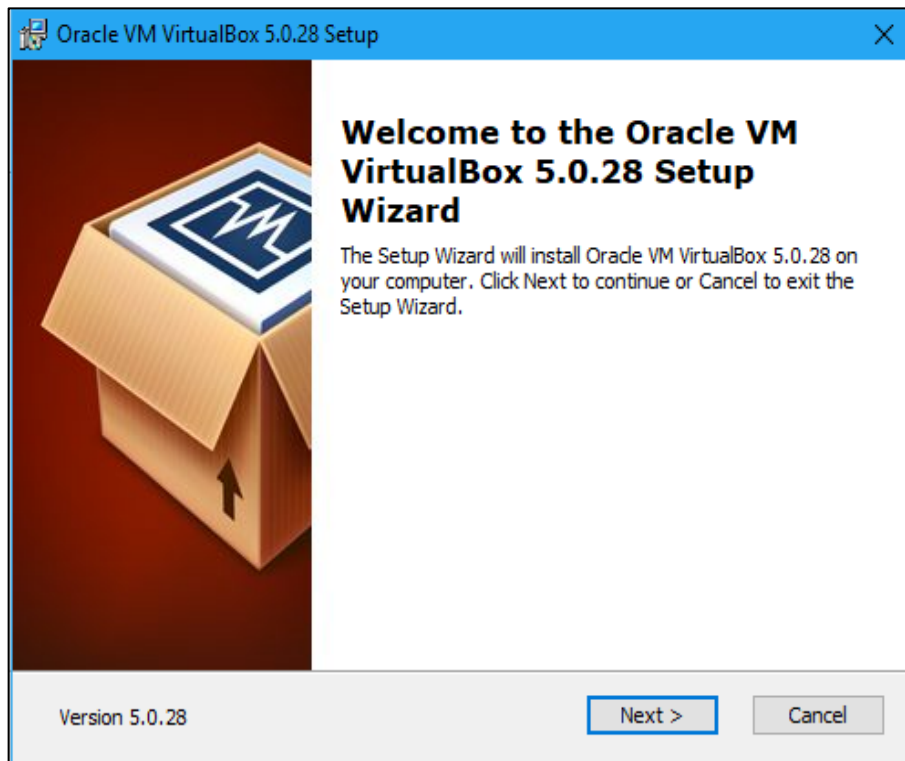
6. Click Install to start the installation.



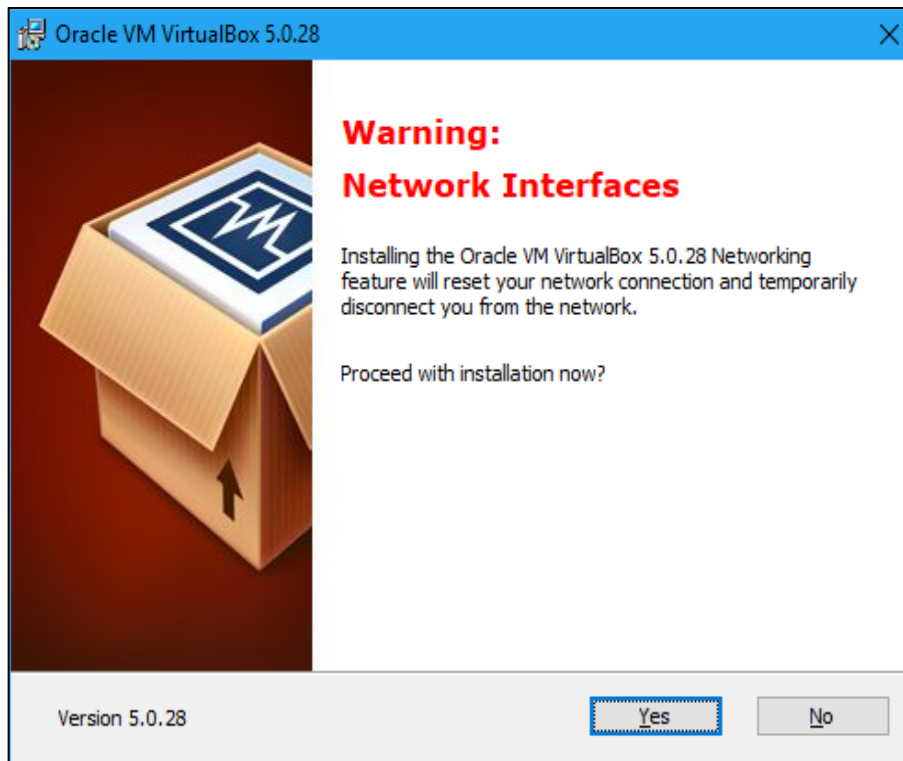
7. It will start to download.



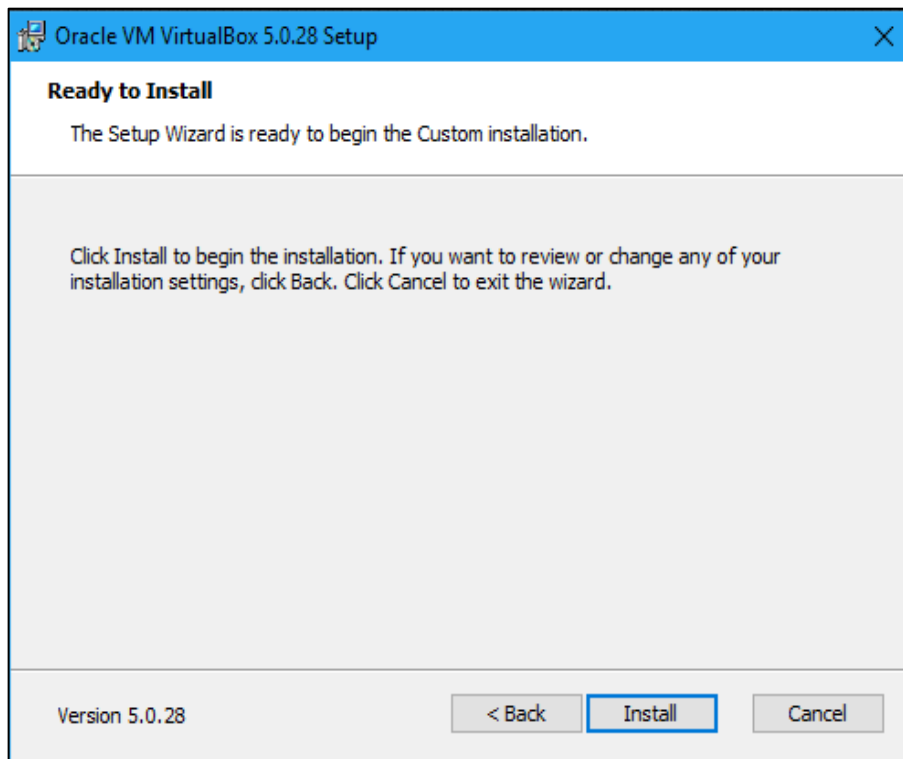
8. If the user has virtual box installed already, then it should be compatible with the genymotion. If the user does not have virtual box, the setup will automatically install with the genymotion. In this case, the virtual box is being installed along with genymotion.
9. Click Next in the Oracle VM VirtualBox setup dialog box.



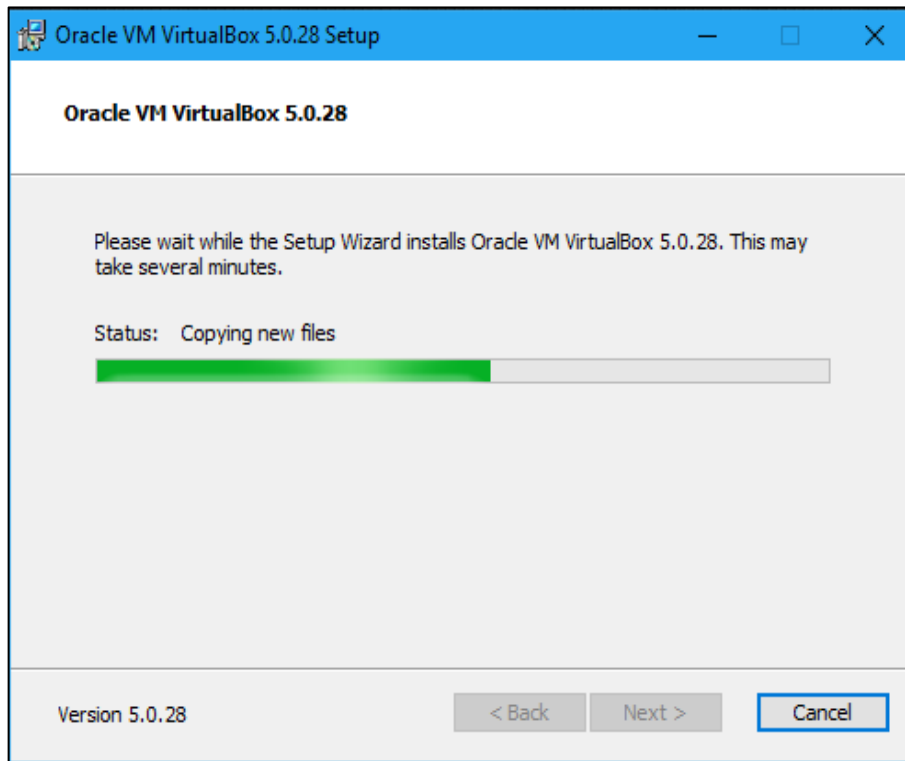
10. Click Yes to install the Oracle VM VirtualBox.



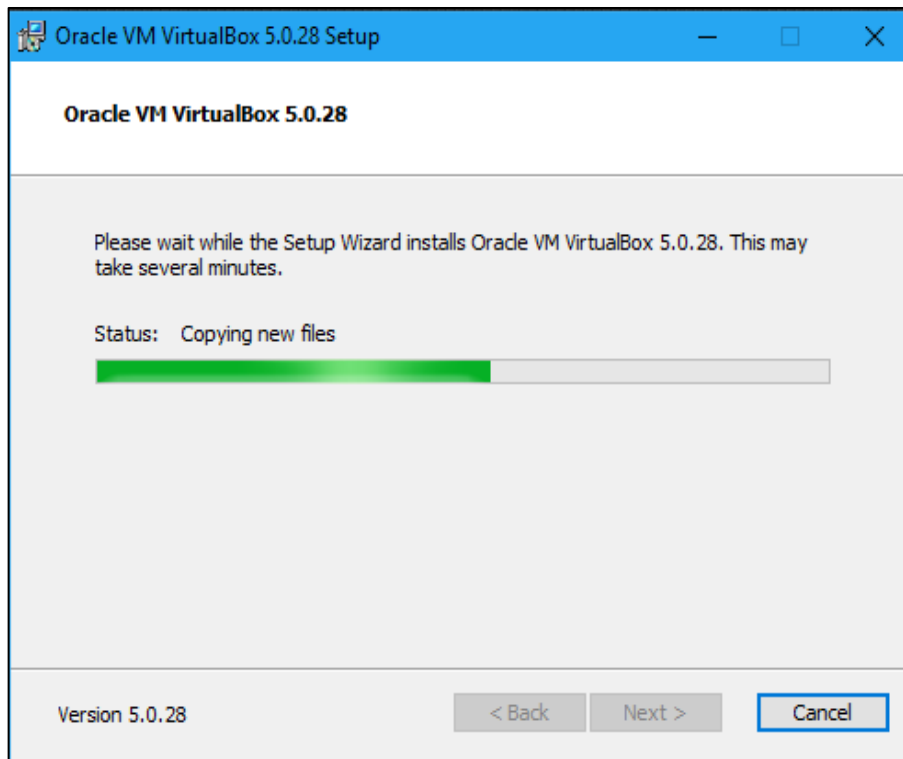
11. Click Install.



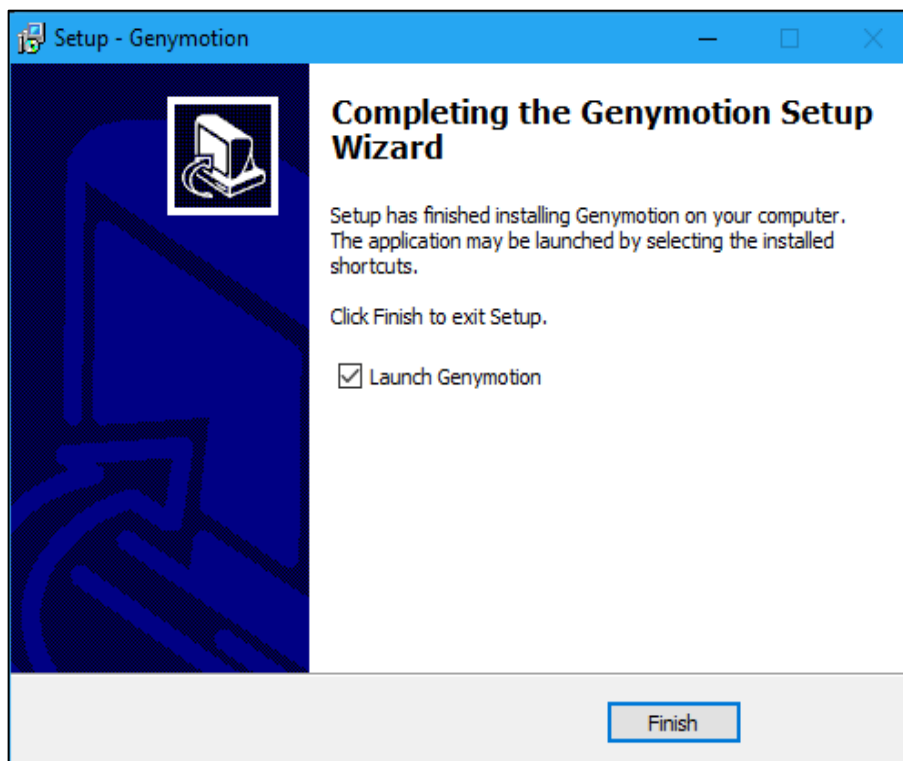
12. The installation will start.



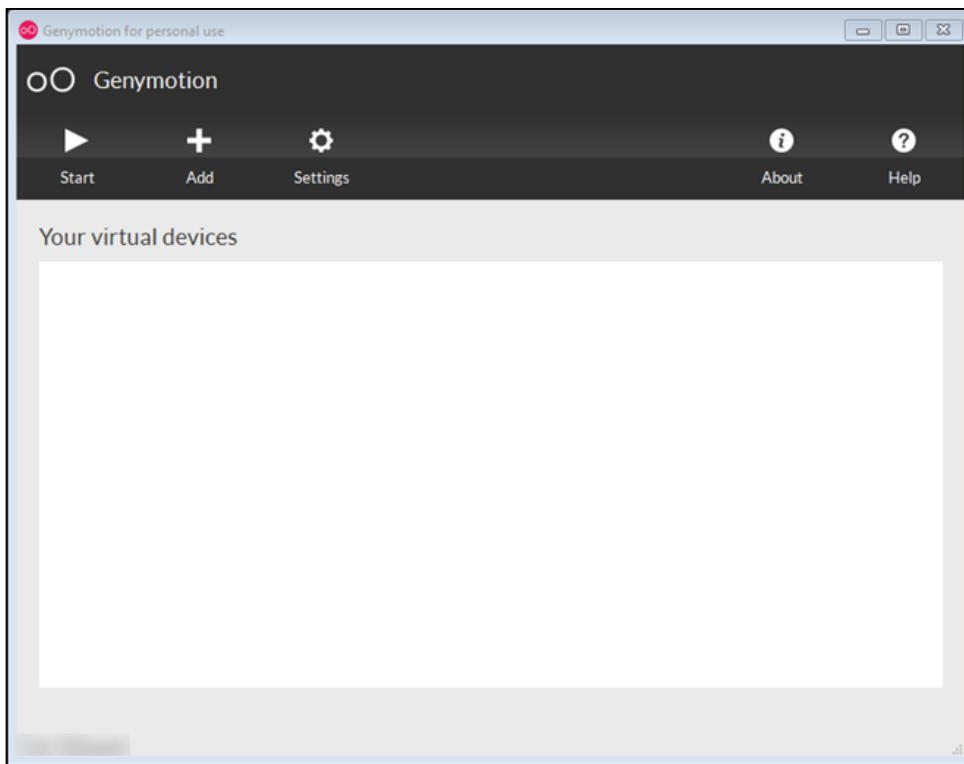
13. After installation click Finish by unchecking the checkbox.



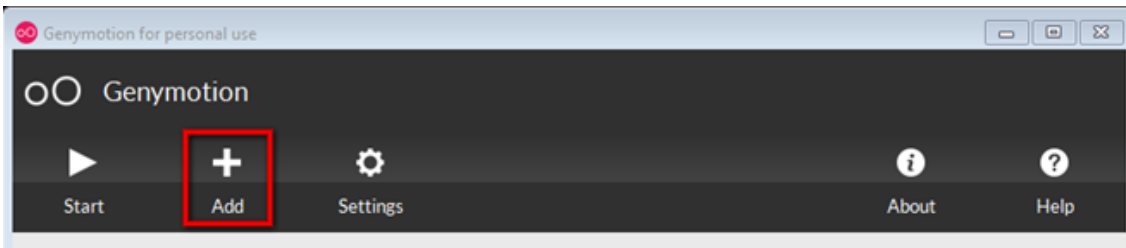
14. Once the installation of VirtualBox is done, click Finish in the genymotion.



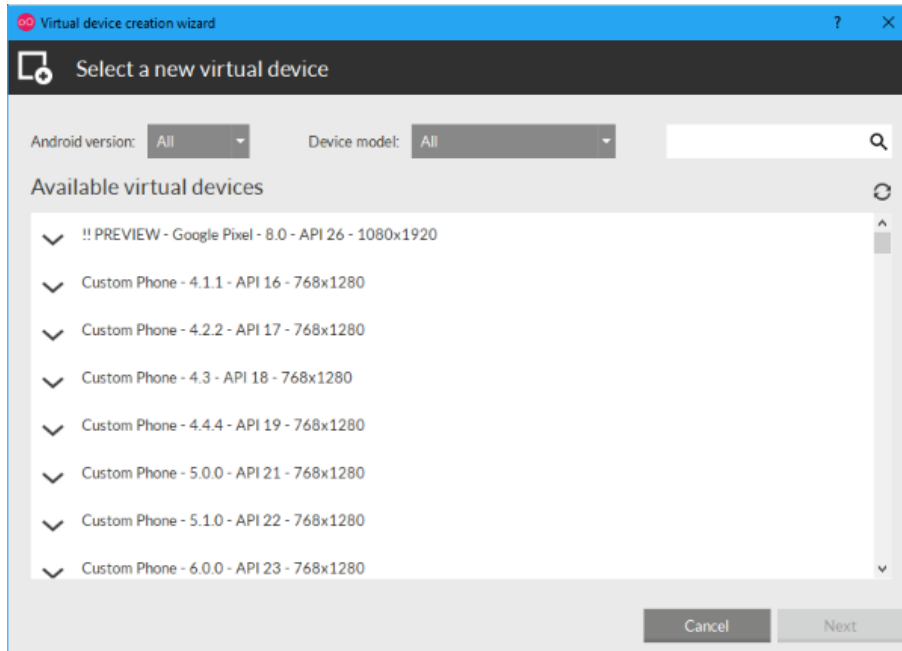
15. The Genymotion will start automatically after the installation.



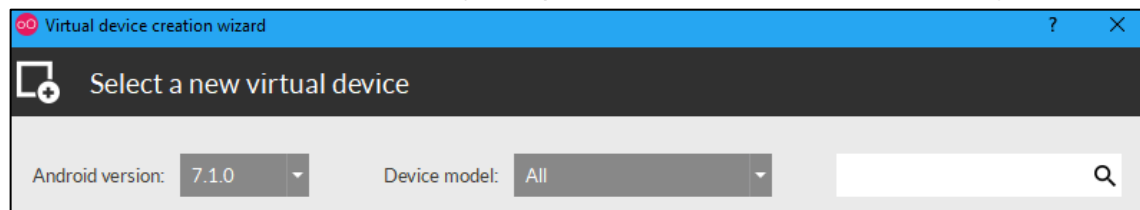
16. After starting the Genymotion, Click Add to add a new virtual device.



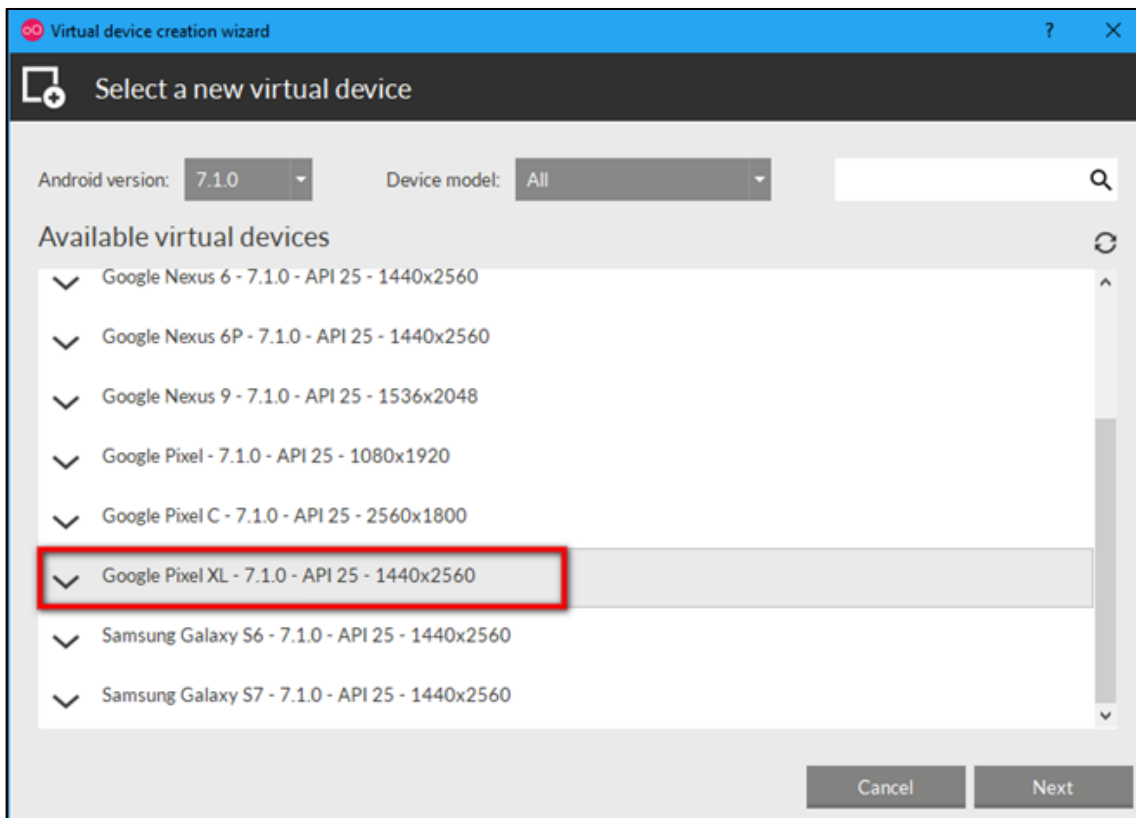
17. On clicking of the Add button, a dialog box appears with the list of different android versions and android virtual devices.



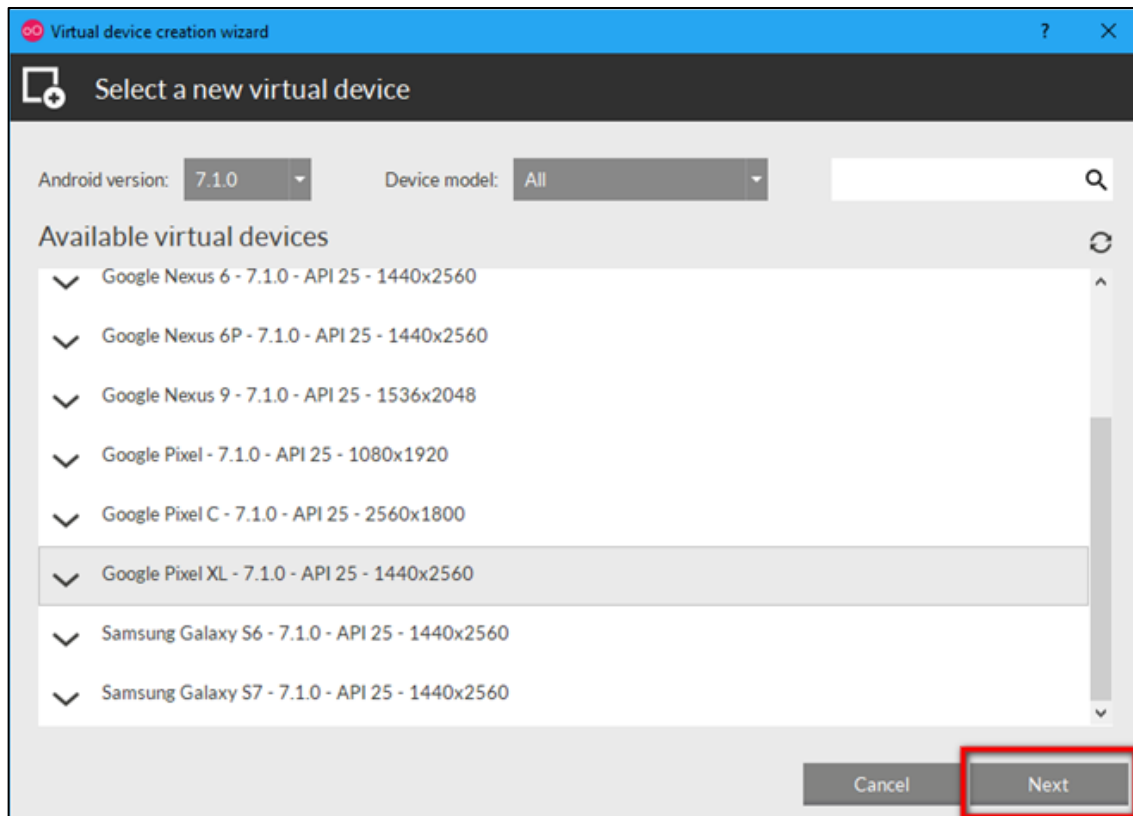
18. From the Android Version dropdown, select the version based on the android version that was downloaded in the Android studio. (Example shows for android version 7.1.0).



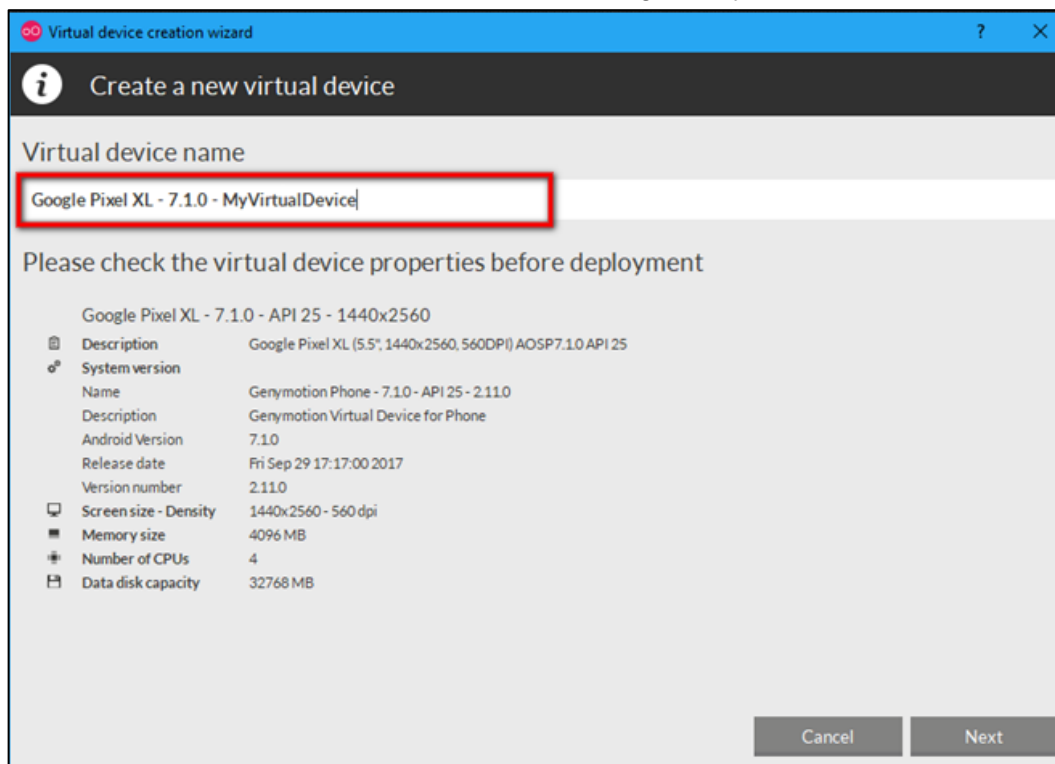
19. Select the device that the user wants to download. Example shows Google Pixel XL.



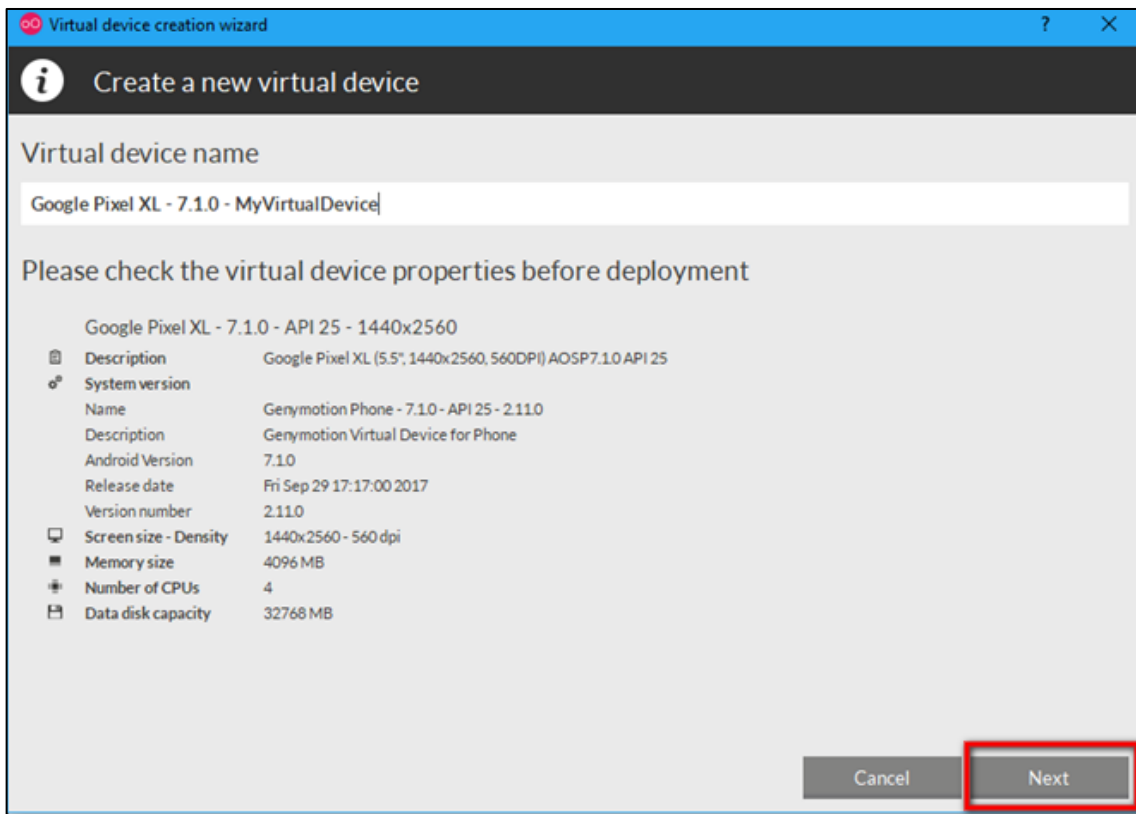
20. After selecting the virtual device, Click Next.



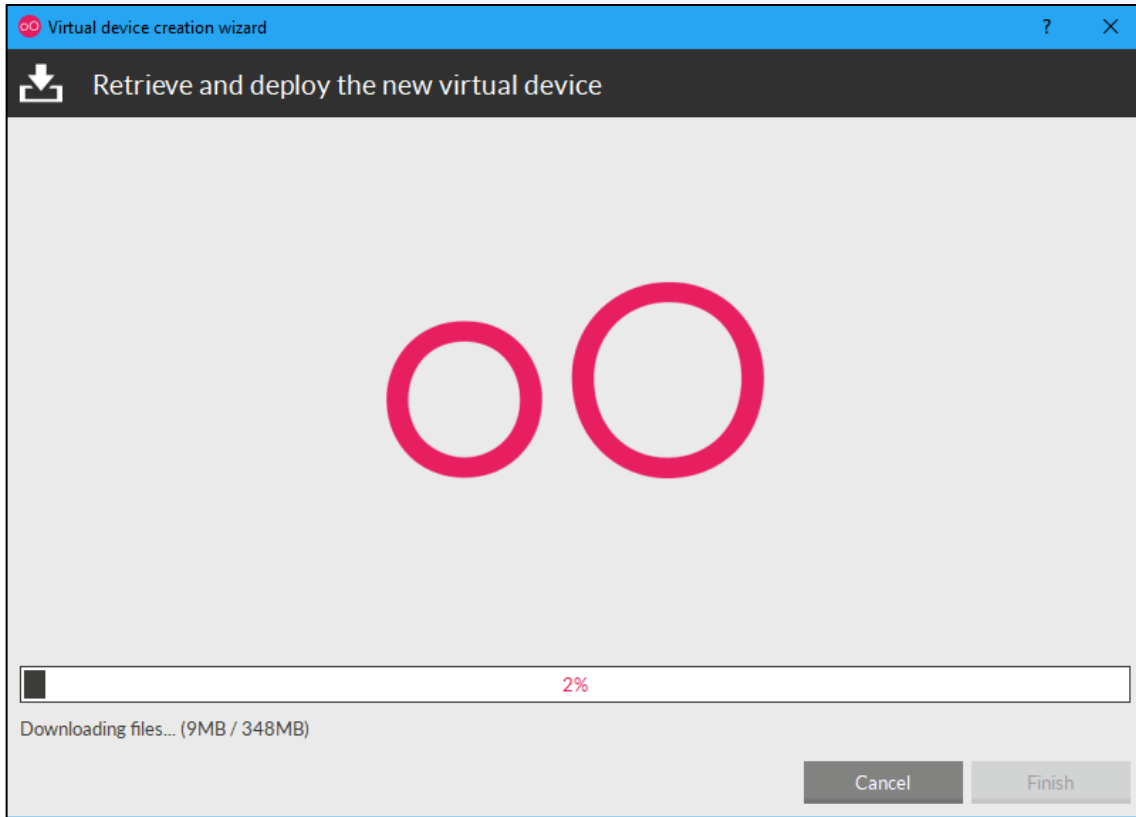
21. Give a name for the virtual device. The user can give any name to the device.



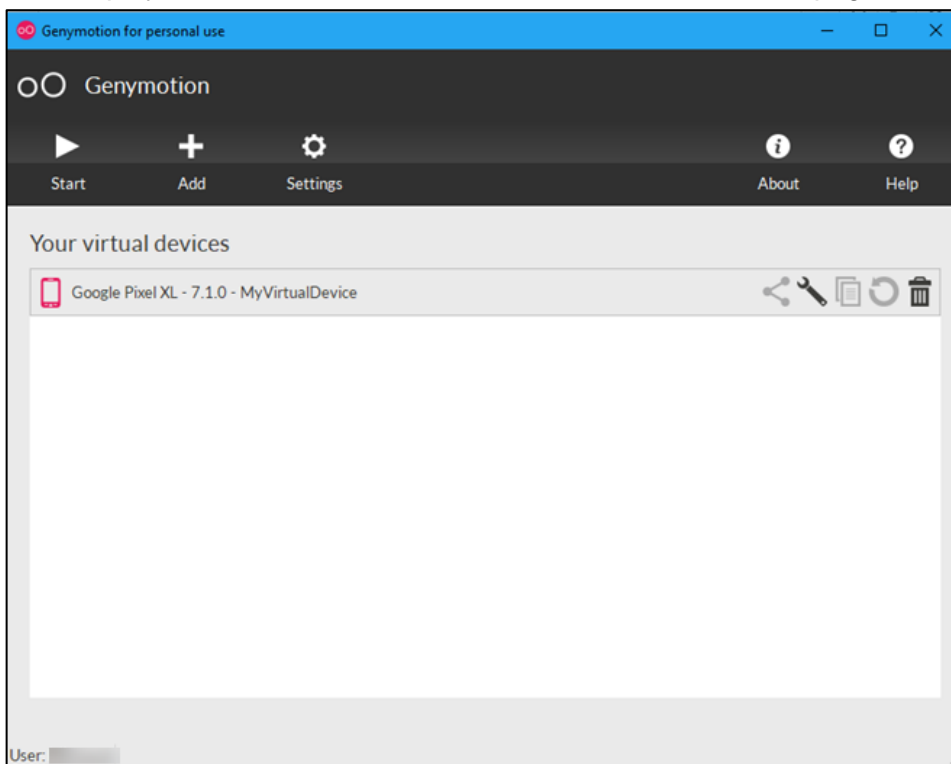
22. Click Next, after which the device will start to deploy.



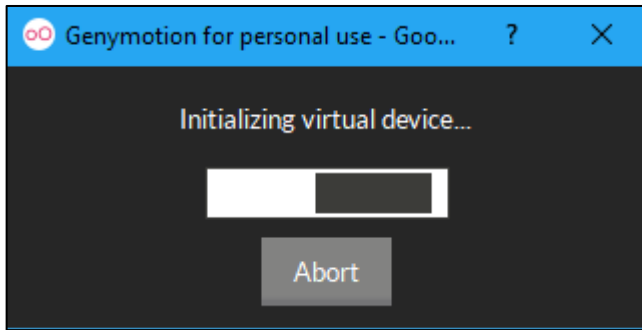
23. Once deployment is done, click Finish.



24. After deployment, the virtual device will be shown in the home page under Your virtual devices.



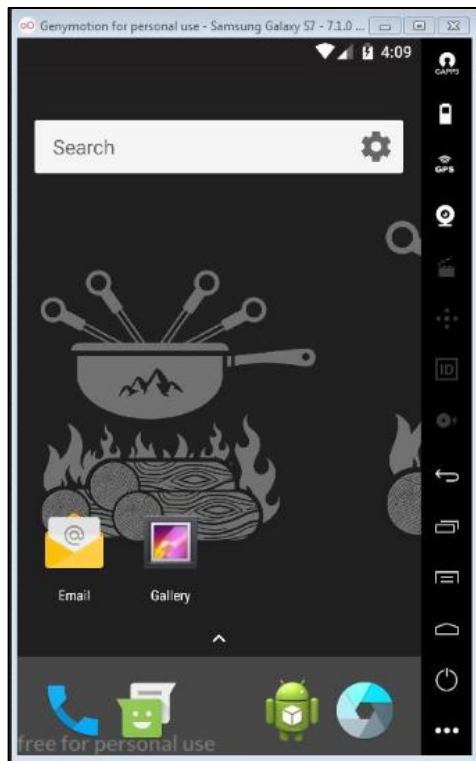
25. Click on the virtual device to start the execution.



26. It will start the virtual device. It may take some time.

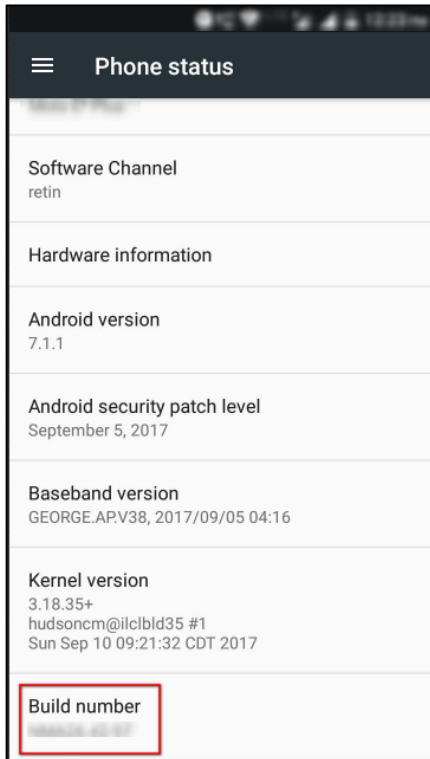


27. Once it starts, the user can perform the steps.

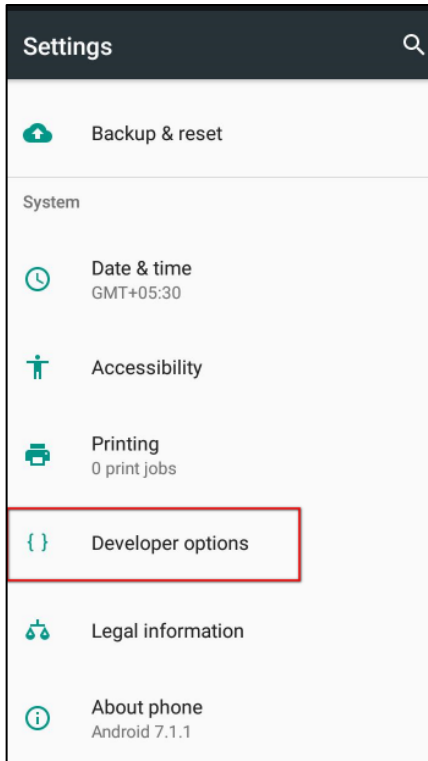


11. SETTING THE PHONE

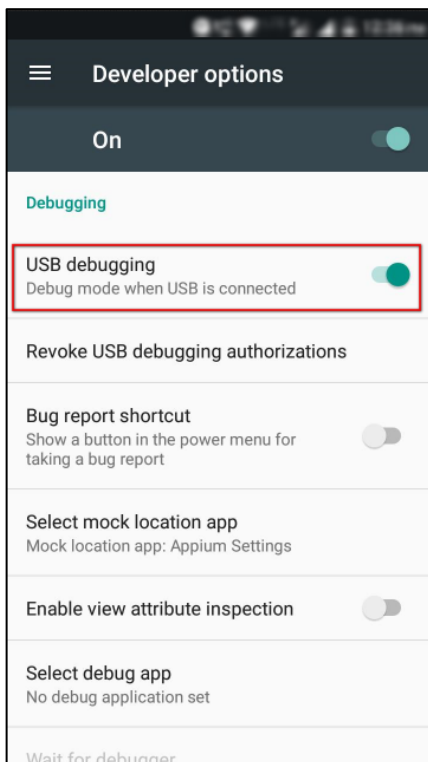
1. Firstly, the user should connect the phone through the USB cable.
2. In the settings of the phone, the user has to enable the developer option.
3. To enable the developer option, open Settings > About Phone > tap on the **Build Number** for 6-7 times, it will display "Now you are a developer".



4. Now go back to settings of phone, the user will find the **Developer Options** setting.



5. In the Developer Options, scroll down and switch on the **USB debugging**.



6. After enabling the Developer Options, it will ask for permission for USB debugging. Click Ok. If the user wants the system to remember, then tap on **Always allow from this computer**.



7. Open any app in the phone for which the test case is to be developed.
8. Now open a command prompt.
9. Enter the command “adb devices” and press Enter. It will show the devices that are connected to the system.

```
Microsoft Windows [Version 10.0.16299.192]
(c) 2017 Microsoft Corporation. All rights reserved.

C:\Windows\System32>adb devices
List of devices attached
ZH33L2HS2L    device

C:\Windows\System32>
```

10. Now enter a command “adb shell” and press Enter. It will open a shell.

```
Microsoft Windows [Version 10.0.16299.192]
(c) 2017 Microsoft Corporation. All rights reserved.

C:\Windows\System32>adb devices
List of devices attached
ZH33L2HS2L      device

C:\Windows\System32>adb shell
nicklaus_f:/ $
```

11. In the shell, enter “`dumpsys window windows | grep -E 'mCurrentFocus | mFocusApp'`” and press Enter.

```
Microsoft Windows [Version 10.0.16299.192]
(c) 2017 Microsoft Corporation. All rights reserved.

C:\Windows\System32>adb shell
nicklaus_f:/ $ dumsys window windows | grep -E 96mCurrentFocus|mFocusApp39
```

12. It will show the `appPackage` and `appActivity`.

```
C:\Windows\System32>adb shell
nicklaus_f:/ $ dumsys window windows | grep -E 'mCurrentFocus|mFocusApp>
mCurrentFocus=Window{8d40d04 u0 com.google.android.calculator/com.android.calculator2
.Calculator}
nicklaus_f:/ $ _
```

13. The values for **appPackage** and **appActivity** will be different based on the app and the android device being used. The user should plug in the specific device that will be used for execution to fetch the above values.
14. For example, if the user opens Calculator app and enters the above command then it will display “`mCurrentFocus=Window{8e7b813 u0 com.google.android.calculator/com.android.calculator2.Calculator}`”
15. In the above message displayed, “`com.google.android.calculator`” is the **appPackage** and “`com.android.calculator2.Calculator`” is the **appActivity**.
16. The `appPackage` and `appActivity` will be used in the writing of Appium code.
17. After writing the code, save the file based on the above-mentioned naming conventions with a file extension of `.py`.