Cookbook

Creating the GeoStack
Workshop VM
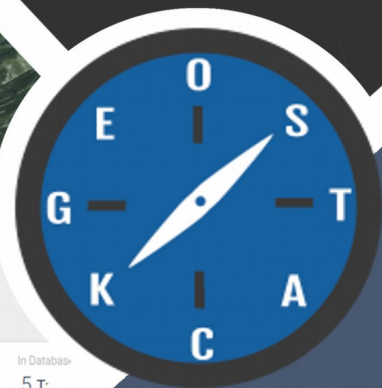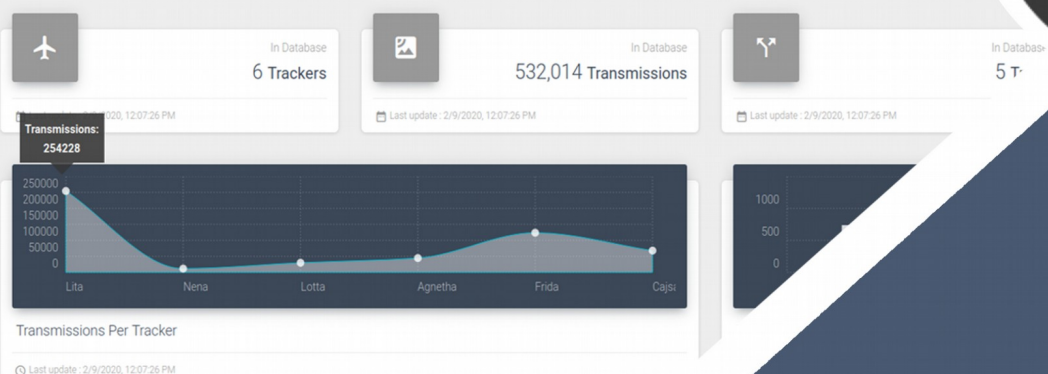
Version : 1.0

Date : 24-06-2020

Author : The GeoStack Project

Purpose of this document

This cookbook serves as manual for installing the GeoStack workshop Virtual Machine.

The GeoStack workshop is the first step in the Full GeoStack course. In the workshop you will learn the basics about most of the components used in the complete Geostack Course.

At the end of this workshop you will have knowledge on the following subjects:

1) Data analyses and processing.

2) Modeling and importing data in a MongoDB data store.

3) Querying data using a Flask-API .

4) Creating a Tileserver used for generating base maps using OpenStreetMap data.

5) Creating a NGINX web server behind which the Geostack components will run.

6) Creating a web application for visualizing Geo spatial data on a 2D map from OpenStreetMap using the Geo spatial framework OpenLayers.

As mentioned before, the workshop is an introduction to the Beginner Course Open Source Geo spatial Programming. During the complete course you will learn much more about the subjects mentioned above and among other things subjects such as:

➔ Using a PostgreSQL data store to generate your own geographical maps.

➔ Creating a secure NGINX web server using ModSecurity .

➔ Creating a Dataset Dashboard.

➔ Creating a fully functional 2D Map Viewer using OpenLayers.

➔ Creating a 3D Map Viewer using Cesium.

➔ Creating Docker-containers for each of the Geostack Components.

NOTE: During this cookbook some commands described are multi line commands. For example the command, used to extract the World Port Index dataset, shown in the illustration below:

```
unzip ~/Downloads/WPI_Shapefile.zip -d
~/GeoStack-Course/Course-Datasets/SHP/World-Port-Index
```

Is actually written as follows:

```
unzip ~/Downloads/WPI_Shapefile.zip -d ~/GeoStack-Course/Course-Datasets/SHP/World-Port-Index
```

Prerequisite for the Workshop are as follows:

➢ **A laptop with an Active Internet network-connection.**

➢ **25GB of free space on you hard drive.**

We are going to start of by installing VirtualBox on your host system. Once you have VirtualBox installed we are going to create a new Ubuntu Virtual machine in which we are going to install the software and tools required for the workshop using the installation scripts provided in the Github Repository called: "GeoStack-Workshop". We are going to clone this repository after the Workshop Virtual Machine is created.

Table of Contents

1. Installing VirtualBox

VirtualBox can be installed on both Windows and Linux. In this chapter I will show you how to install the VirtualBox software on both operating systems. Afterwards we will download and install the VirtualBox extension pack. The extension pack extends the functionality of VirtualBox base packages.

1.1 Installing VirtualBox on Windows

To be able to run VirtualBox on Windows you first have to make sure that:

➔ Virtualization technology VTx support is turned **ON** in the BIOS of your system. Read the manual of your BIOS to find this setting.

➔ Hyper-V support is turned **OFF** in the Windows Features settings of your Windows Host Operating System.

NOTE: Hyper-V is required for Docker to run on your HOST system. If you have Docker running on your system VirtualBox will not work. Docker and VirtualBox can not run at the same time. So Docker uses a Hyper-V VM to run on Windows and VirtualBox uses another VM sandbox system that both need the VTx functions from the hardware and as these VTx functions can not be shared the user has to turn Windows Hyper-V support on or off depending on what software to run on the Windows Host system.

To turn off Hyper-V in your Windows system you have to perform the following steps:

1) Press the Windows key + X and select Apps and Features
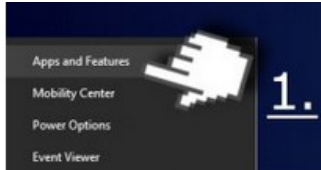
2) Scroll down to the bottom and click Programs and Features
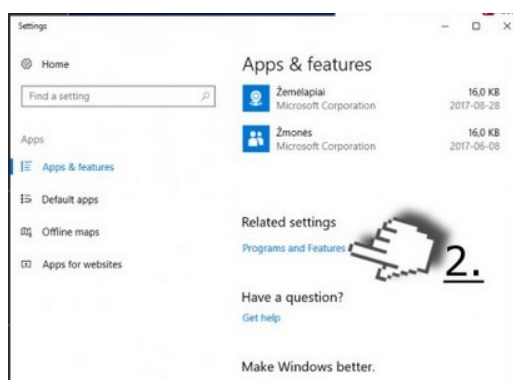
3) Then click on: Turn Windows features on or off on the left

4) Locate Hyper-V in the screen that pops up and uncheck it.

5) Click on OK to save the changes and reboot the system.

NOTE: this option will not uninstall Hyper-V, but disable the feature temporarily. It will be enabled automatically after the installation of crucial Windows Updates.

Source: https://ugetfix.com/ask/how-to-disable-hyper-v-in-windows-10/

Now that we have disabled Hyper-V we can start with the installation process of VirtualBox by performing the following steps:

1) First we need to download the VirtualBox executable. This is can be downloaded via the following link: https://www.virtualbox.org/wiki/Downloads

2) Select Windows hosts under the section: "VirtualBox binaries". This will open a download pop-up. Click on save file.
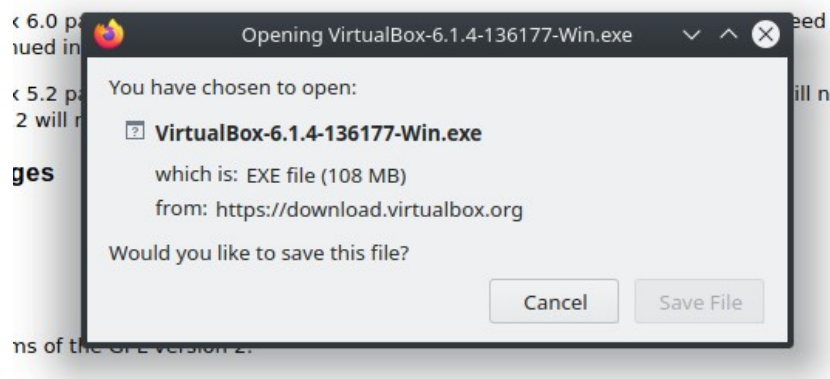
3) When the executable is finished downloading you should click on the downloaded .exe file.

4) When you start the installation, you should get a prompt like the one below.
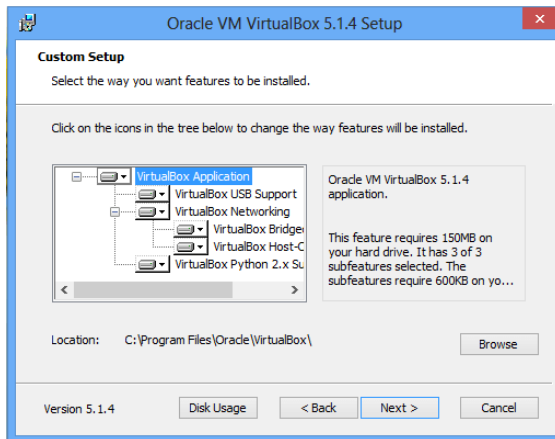
5) During the installation wizard, you'll get a Network Interfaces warning. Click Yes to proceed.

6) On the next screen, click Install to begin installing the program.

7) When prompted with a message to install (Trust) Oracle Universal Serial Bus, click Install to continue and wait for the installation process to finish.

1.2 Installing VirtualBox on Linux

The process for installing VirtualBox in Linux is a lot easier than on Windows. You can choose whether you want to use the version provided by the Ubuntu package repository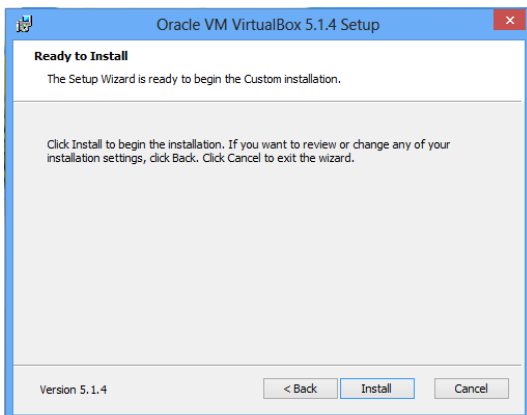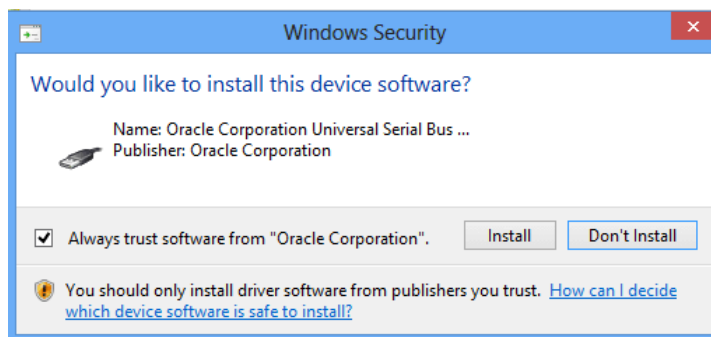. This is VirtualBox version 5.X. You can also install the latest version of VirtualBox which is version 6.X. This is done by adding a Personal Package Archive (PPA) repository from the VirtualBox Project to the systems software repository list. This enables the system to install the latest VirtualBox version and check for updates for this version.

Both installing VirtualBox V5 and V6 are described below.

1.2.1 Installing VirtualBox version 5.X

The installation of VirtualBox 5.X is done by opening a Terminal program (Ctrl + Alt + t) and running the following commands:

1) Update the local package database.

   sudo apt update

2) Upgrade the local package database.

   sudo apt upgrade

3) Install the VirtualBox package

   sudo apt install virtualbox

4) Open VirtualBox by entering the command: virtualbox in the terminal or by simply clicking on the shortcut which was created.

That's it! Now we first need to install the VirtualBox extension pack as explained in section 1.3 below and then we can create a new Virtual Machine as explained in chapter 2.

1.2.2 Installing VirtualBox version 6.X

The installation of VirtualBox 6.X is done by opening a Terminal program (Ctrl + Alt + t) and running the following commands:

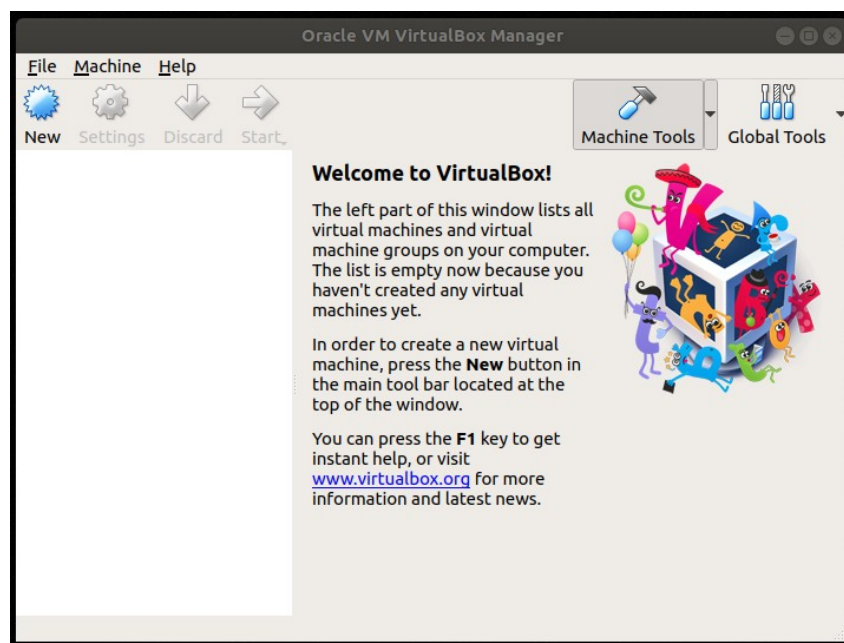1) Update the local package database.

    `sudo apt update`

2) Upgrade the local package database.

    `sudo apt upgrade`

3) Add the signing keys from the VirtualBox website.

    `wget -q https://www.virtualbox.org/download/oracle_vbox_2016.asc -O- | sudo apt-key add -`

    `wget -q https://www.virtualbox.org/download/oracle_vbox.asc -O- | sudo apt-key add -`

4) Add the repository to the repositories list.

    `sudo add-apt-repository "deb [arch=amd64] http://download.virtualbox.org/virtualbox/debian $(lsb_release -cs) contrib"`

5) Update the local database again.

    `sudo apt update`

6) Install the VirtualBox package

    `sudo apt install virtualbox-6.0`

7) Open VirtualBox by entering the command: virtualbox in the terminal or by simply clicking on the shortcut which was created.

That's it! Now we first need to install the VirtualBox extension pack as explained in section 1.3 below and then we can create a new Virtual Machine as explained in chapter 2.

1.3 Installing the VirtualBox Extension Pack

Installing the VirtualBox Extension Pack allows you to unlock advanced VirtualBox features. The VirtualBox extension pack adds support for USB 2.0 and USB 3.0 devices, VirtualBox RDP, disk encryption, NVMe and PXE boot for Intel cards. For more information related to the VirtualBox extension pack you should visit the URL: https://www.virtualbox.org/manual/ch01.html#intro-installing

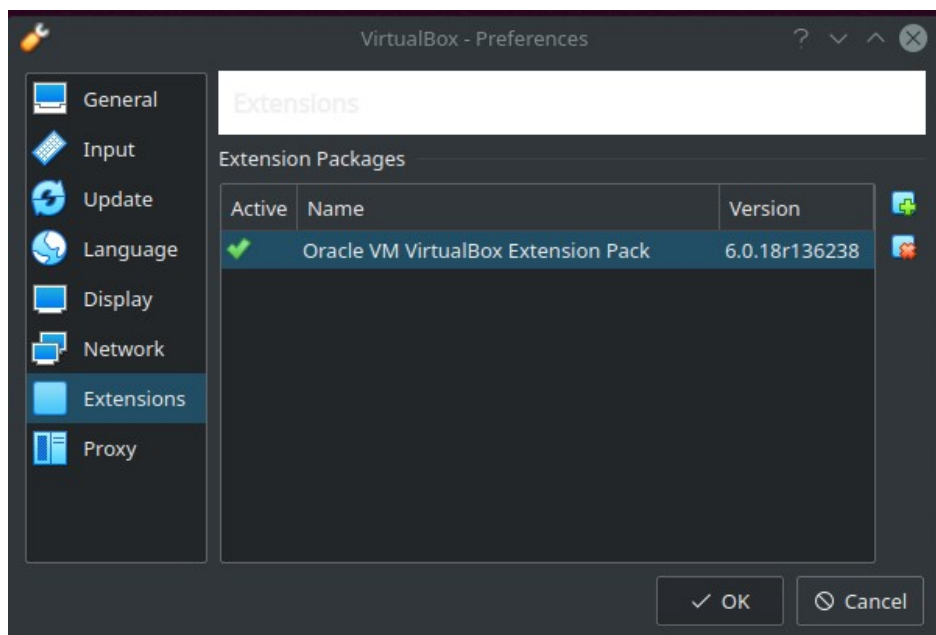Installing the VirtualBox Extension Pack can be done in 2 ways:

➔ The first way is by downloading and installing it via the terminal. If you have installed **VirtualBox Version 5.X in Linux** it's recommended to install the extension pack in this way.

➔ The second way is by downloading the extension pack from the official VirtualBox website. If you installed **Version 6.X in Linux** it's recommended to install the extension pack in this way.

1.3.1 Using the terminal

When installing VirtualBox 5.X in Linux you need to install 2 extra packages. These packages are as follows:

➔ The VirtualBox extension pack by using the command: sudo apt install virtualbox-ext-pack

➔ The VirtualBox Guest Additions ISO by using the command:

sudo apt install virtualbox-guest-additions-iso

If the installation process is finished you can go to File → Preferences → Extensions in your VirtualBox window and you should see something similar to the illustration below.


NOTE: Your version can be different so don't worry about that!

1.3.2 Downloading from the website

First you need to download the VirtualBox extension pack. We do this by going to the following URL: https://www.virtualbox.org/wiki/Downloads.

Beware to download the version of the extension pack which is the same as the version of your VirtualBox software. Check the exact VirtualBox version number by starting the VirtualBox application and then in the Help menu click the menu option 'About VirtualBox'. Now let's start the installation process of the VirtualBox extension pack by performing the following steps:

1) In the VirtualBox start screen go to file → preferences.
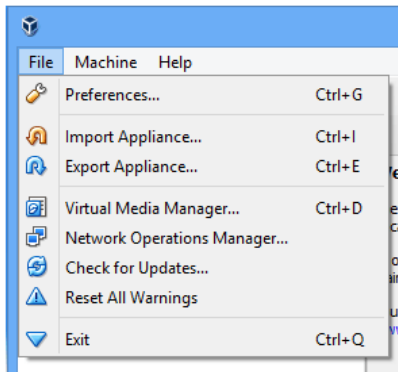
2) Go to extension and then click on the browse button to find the download for the extension pack.
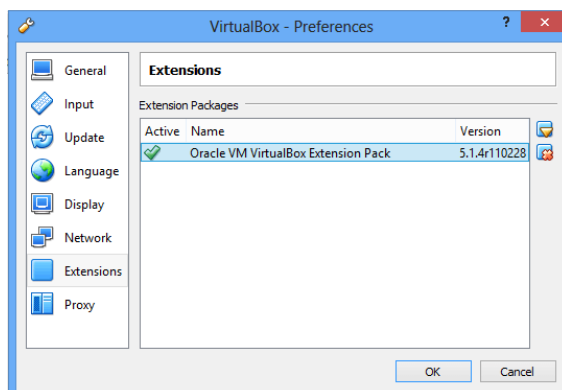
3) When prompted enter the password of the Linux account to authorize the software installation.

NOTE: After entering one character a screen can pop up showing the installation progress bar. You first have to minimize this pop up before entering the rest of the password.

That's it! Now you have VirtualBox with the extension pack running on your system! Now we have to create a new Virtual Machine and install Ubuntu.

1.4 Downloading an Ubuntu ISO

The Beginners Course Open Source GeoSpatial Programming for Data Scientists was created using Ubuntu 18.04. As of April 16, 2020 Ubuntu 20.04 LTS (Long Term Support) has been released. This version has been tested and can also be used during The Beginners Course Open Source GeoSpatial Programming for Data Scientists although the illustrations used in the documents are created using Ubuntu 18.04.

Before we can continue the installation process of the Virtual Machine we first need to download an Ubuntu ISO form the website. This can be done by clicking on the following URL: https://ubuntu.com/download/desktop

When on the website you should get the option to download the Long Term Support Ubuntu version which is version 20.04 as of April 16, 2020. This can be done by clicking on the download button as shown in the illustration below.

You also have the option to download the previous LTS version which is version 18.04. This can be done by navigating to the following URL:

https://ubuntu.com/download/alternative-downloads

The scroll down until you see the screen as shown in the illustration below and then click on the button encircled in red in the illustration below:

NOTE: During this cookbook we are going to use the Ubuntu 18.04 LTS version. You can also use Ubuntu 20.04 if you prefer this version. If you choose 20.04 some steps, described in this cookbook, are a bit different.

2. Installing the GeoStack workshop Virtual Machine

At this point you have VirtualBox on your system. Now we need to install the GeoStack workshop virtual machine. To be able to install the Workshop VM we first need to create a new Virtual machine. After we have created the Virtual Machine we are going to install an Ubuntu ISO file.
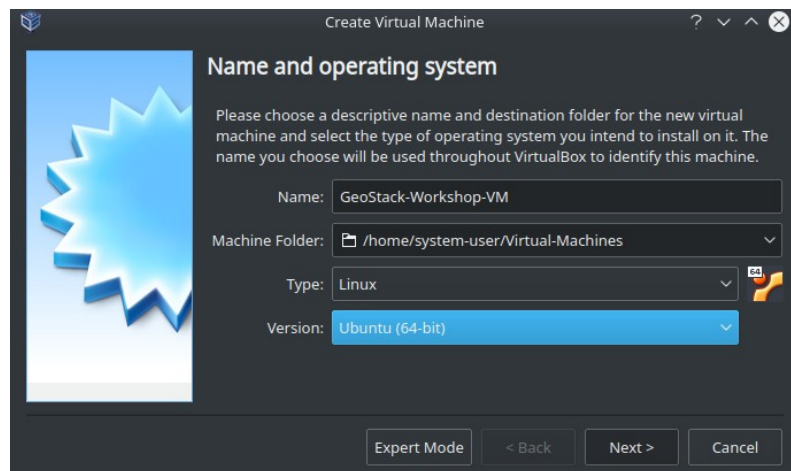
2.1 Creating the Virtual Machine in VirtualBox

Creating a Virtual Machine is done by performing the following steps:

1) Click on the button: "New" in the VirtualBox start screen.
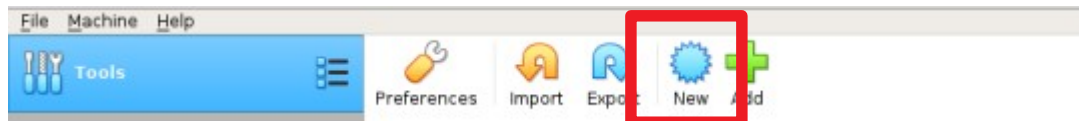
2) Give the Virtual Machine a suitable name in this case we give it the name: "GeoStack-Workshop-VM".

3) Assign the amount of RAM you want the machine to use. The minimal amount of RAM is 1 GB (1024MB) for a VM. If your PC has enough RAM the advice is to assign 3 GB (3072 MB) or 4 GB (4096 MB) RAM because Ubuntu will use up to 2 GB (2048 MB) already for itself.
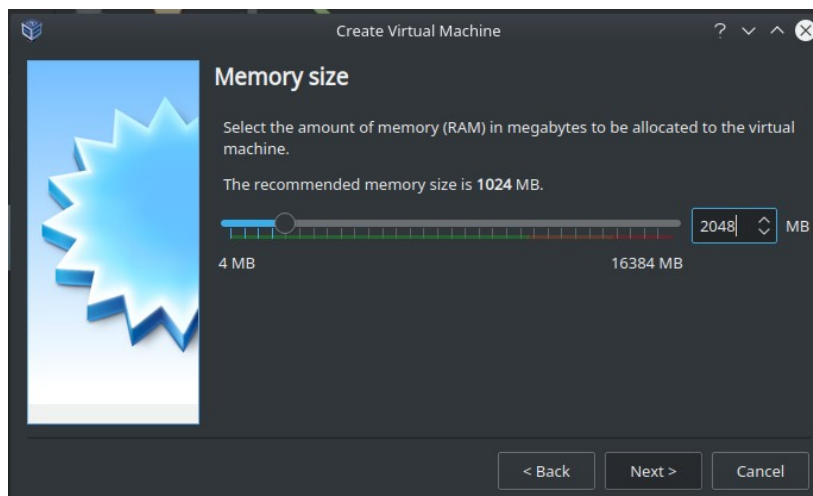
4) Select: "Create a virtual hard disk now" and press create.
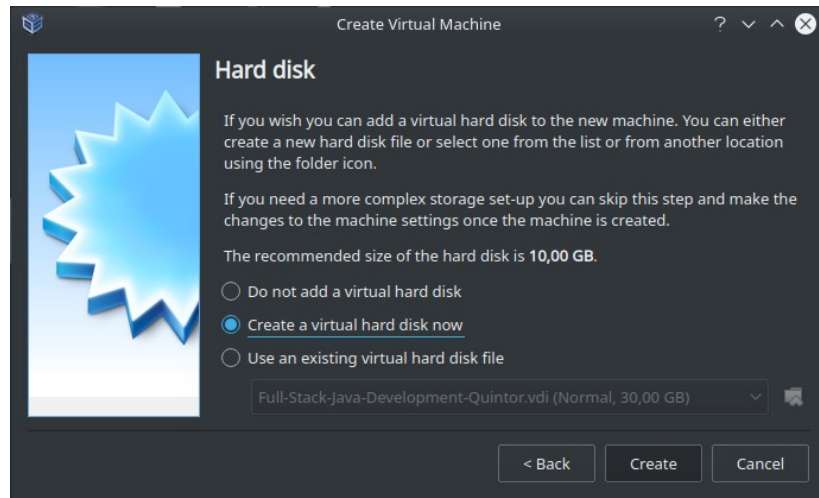
5) Set hard disk file type to VDI (Virtual Disk Image).
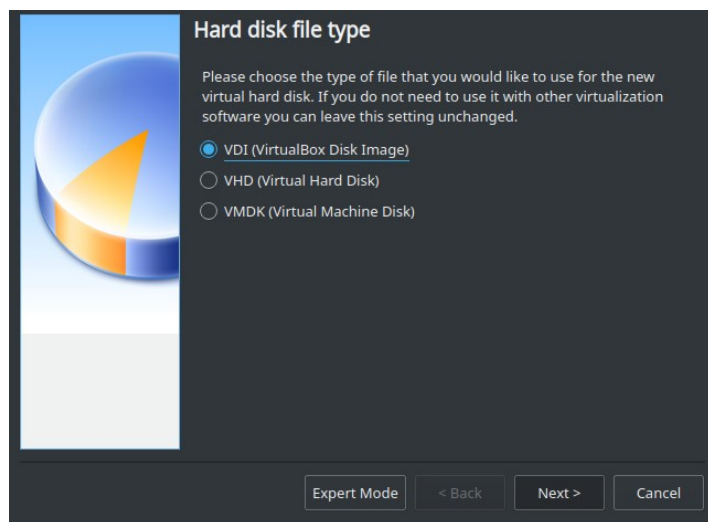
6) Select allocate disk space dynamically (Dynamically allocated). Tip: choose fixed disk size if performance matters to you.

7) Select the location where you want to store the Virtual Disk Image file (.vdi file). In the perspective of managing your Virtual Machines, it's useful to create a centralized folder on your host system in which you create your VM's. The folder name could for example be called: " VirtualBox_VMs". This folder can also be used to store files such as an ISO file and the VirtualBox Extension Pack file. Then set the size of the hard disk to a minimum of 20 GB. The reason we are choosing to set the disk size to 20GB is because increasing the disk size of a VDI file can be a tricky process. Finally click the Create button to create the VDI file.
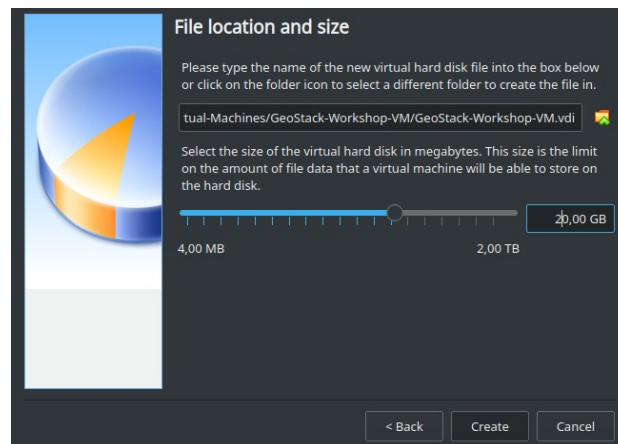
8) Select the Virtual Machine 'GeoStack-Workshop-VM' by clicking on the VM's name so it highlights in blue if it is not already highlighted. Then click the icon with the green "Start" arrow to start the Virtual Machine in VirtualBox.
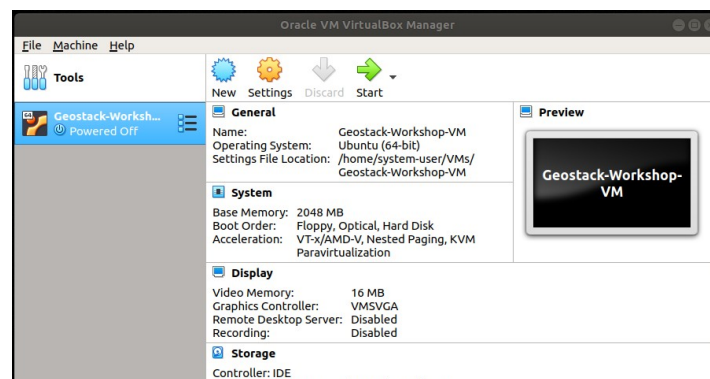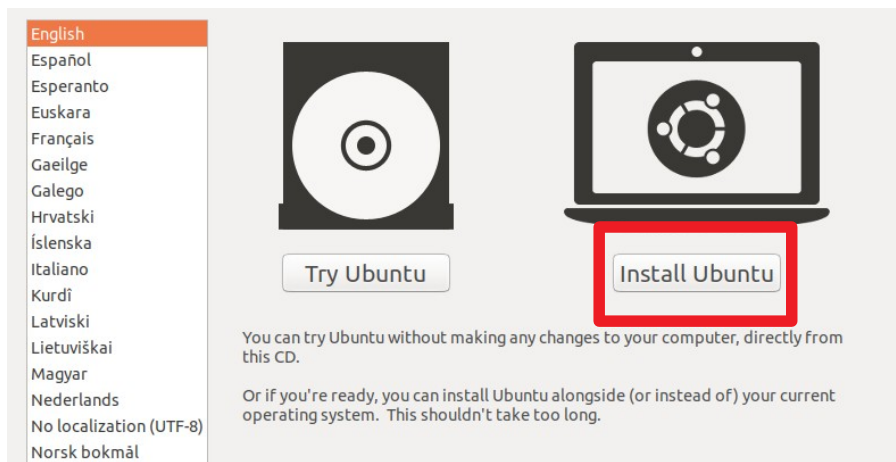

9) Select the Ubuntu ISO file you have downloaded in section 1.3 as the start-up disk for the Virtual Machine by clicking on the yellow folder icon on the right to browse and select the ISO file and then click the Start button to load the ISO file into the Virtual Machine.
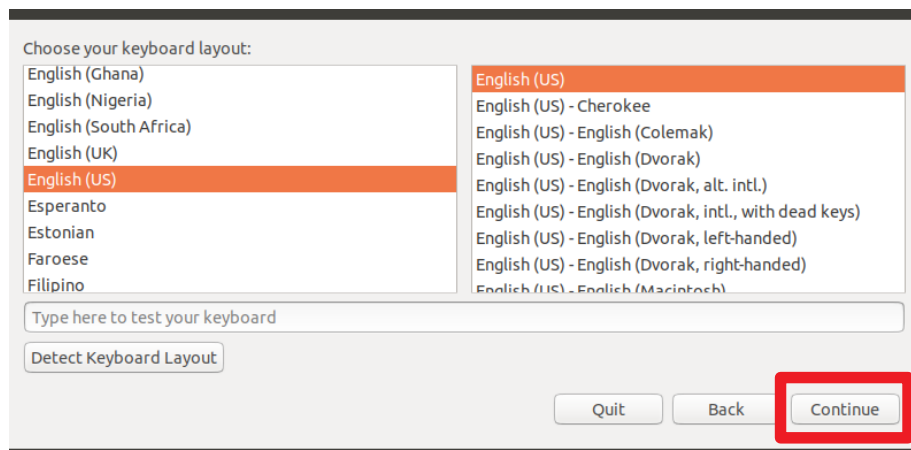
2.2 Installing the Ubuntu ISO

1) Once the ISO file is loaded, click on the button "Install Ubuntu".

2) Select the keyboard layout. This depends on your preferences. For example choose US (English) - English (US, euro on 5) for the Netherlands. Then click on Continue.


3) Select "Minimal installation" which will only install the base software of the Ubuntu Installation. For more information related to a minimal Ubuntu installation you should visit the following URL: https://wiki.ubuntu.com/Minimal.
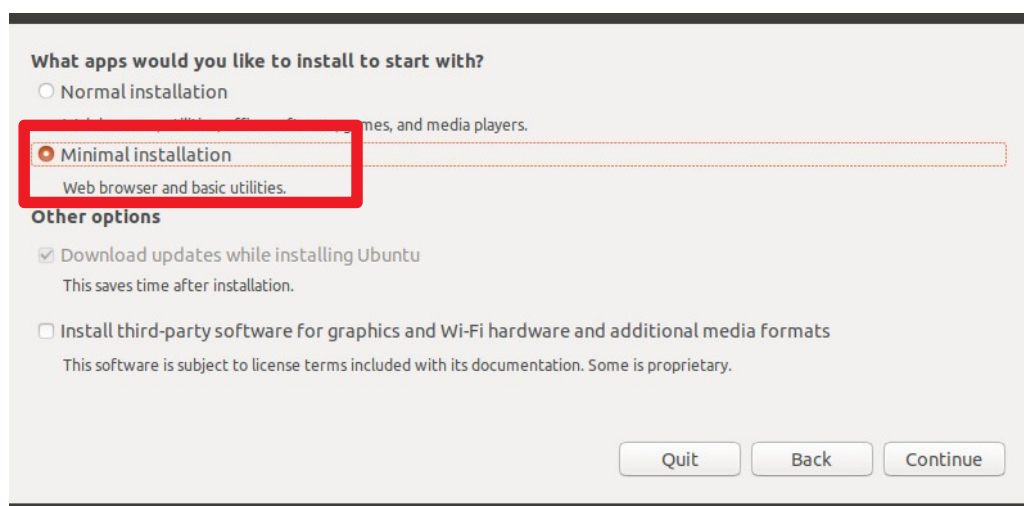
4) Select erase disk and install Ubuntu. NOTE: Erasing the disk will only erase the disk of the Virtual Hard Drive in the Virtual Disk Image file (.vdi file) of the Virtual Machine, NOT the physical disk of your host operating system, so don't worry!
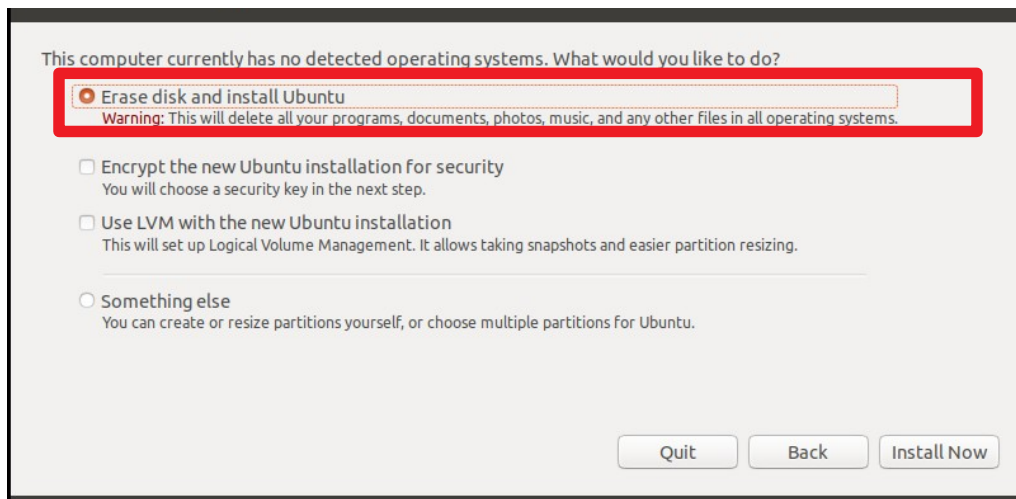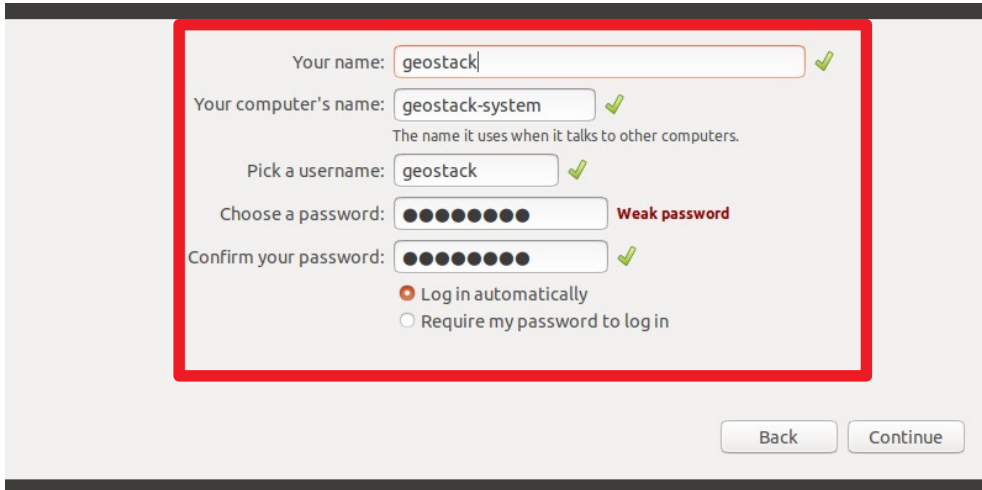

5) Click on continue.
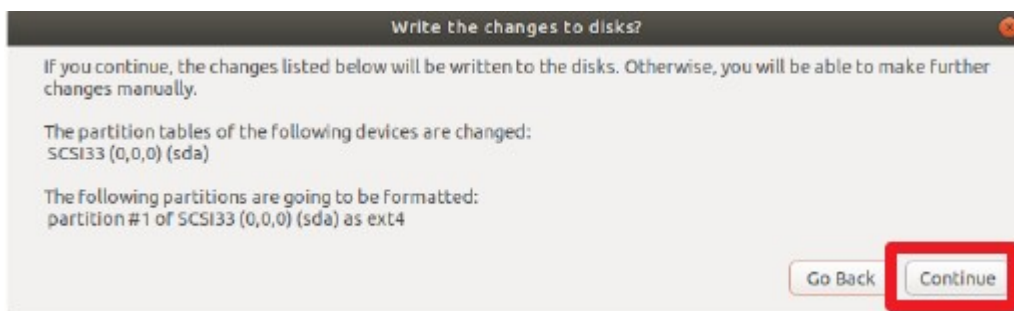
6) Select a location. NOTE: This will define the timezone which will be used in your VM so it's recommended to choose your current location.

7) Enter the same information as in the image below. The field: "Your computer's name" is automatically set to: "geostack-VirtualBox" instead of: "geostack-System". If you want you can keep the default system's name. The password of the VM has to be set to geostack. You can also choose one yourself but then you will have to use that password when you are asked to fill in the password: "geostack" during the workshop. When selecting login automatically, you will not have to login while starting the VM, however when the VM goes to the lock-screen you will still have to enter the password: "geostack". If you choose to use another password you will have to change geostack with the password you chose.


8) Wait until the installation is finished. The process takes about 5 minutes depending on you network-connection. After the installation process is completed 6GB of additional disk space is used.
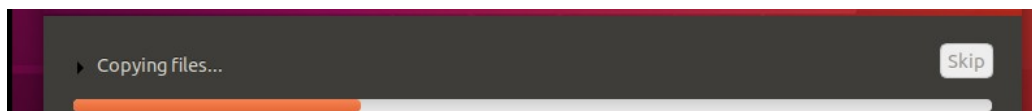
9) Press the button: "Restart Now" which will automatically reboot the Virtual Machine.
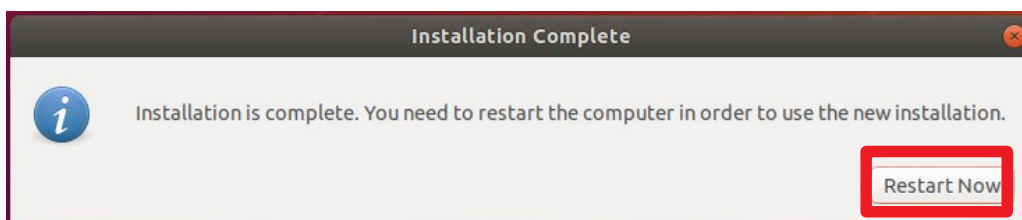
10) When the system is rebooted the following screen will appear, asking you to remove the installation medium. Since this is a Virtual Machine we don't need to remove any installation medium so we can Press the Enter key to continue the reboot process when prompted with the text 'Please remove the installation medium, then press ENTER:'
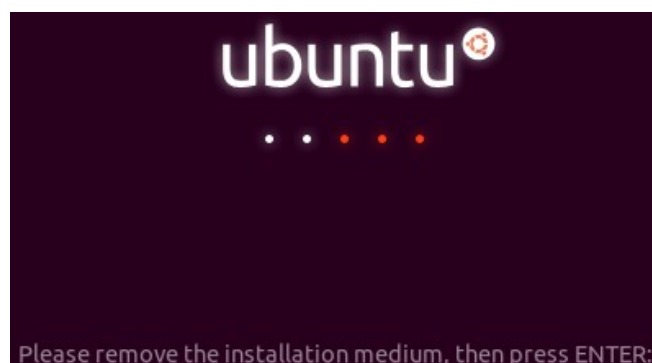
11) After pressing enter you should wait for a few seconds after which you will automatically be logged in to the Ubuntu Virtual Machine. If you want to go back to controlling your host system you can press the Right Ctrl + the Host key on your keyboard. This will open the selection menu at the top of your VirtualBox window.

NOTE: It's possible that the screen stay's black after pressing ENTER on you're keyboard. The reason for this is a bug in the snap server deamon in combination with the Linux kernel mostly in Ubuntu versions earlier than 19.10 as described here: https://askubuntu.com/questions/1147756/a-start-job-is-running-for-wait-until-snapd-is-fully-seeded . There is a workaround for the Ubuntu 18.04 LTS version used in the workshop in 5 simple steps:

➔ **First wait for about 3 - 5 minutes until the blinking 'LED' in the blue disk icon stops blinking.**

➔ **Then in the Machine menu click the menu option ACPI Shutdown. In a flash you see the warning message indicating the bug: A start job is running for wait until snapd is fully seeded.**

➔ **After that restart the Virtual Machine again with the green Start icon.**
   Then install the software updates manually as described in section 4.2: "Manually updating Ubuntu using the CLI" and reboot again.

➔ **If the reboot gives a black screen again then simply shutdown via Machine / ACPI Shutdown for the second time and start the VM again. Now all Power Offs, Shutdowns and Reboots should work normally.**
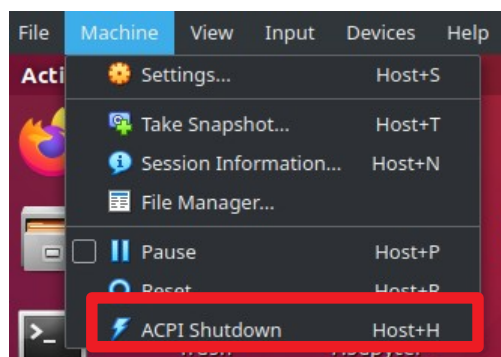

12) Some screens will pop up in the Virtual Machine. In these screens you are able to select settings related to telemetry services, automatic updates and privacy settings. **These screens differ per Ubuntu version**. When selecting the settings you have to make sure that you don't send any system info.

In **Ubuntu 19.10 and 20.04** you should perform the following steps. Click Skip to close the pop-up window Online accounts. Then check No, don't send system info in the next pop-up window Help improve Ubuntu and click Next. In the next screen Welcome to Ubuntu leave the Privacy slider switch for Location Services turned off and click Next. Finally close the pop-up window Ready to go by clicking Done.

13) Sometimes after a little while an Ubuntu Installer screen pops up asking you to install software updates, click the button "Remind Me Later" to skip the installer update. Install the updates of the software packages in the usual way as described in the next step (14). The pop-up screen is shown in the illustration below.
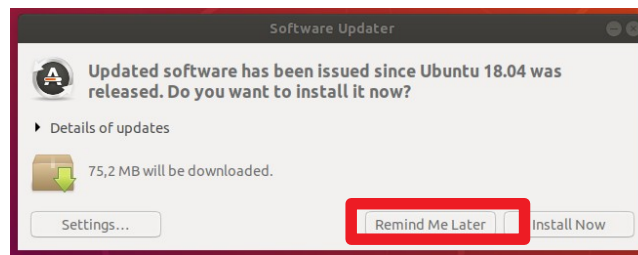
14) Now update the software packages in the usual manual way either on the CLI (Command Line Interface) with the Terminal application or with the Ubuntu Software Updater application.

Go to section 4.2: "Manually updating Ubuntu using the CLI" or section 4.3: "Manually updating Ubuntu using the GUI" to run the update commands.
This is important to learn because Ubuntu automatically checks and shows a notification message if manual software updates need to be run in order to keep the computer up-to-date and safe.

15) OPTIONAL: If you want to disable the Ubuntu lock screen you should read section 4.1: "Disabling the Ubuntu lock screen". By doing this you won't be bothered by the fact the the system automatically locks after 5 minutes.

16) OPTIONAL: In the following chapters you are asked multiple times to open a terminal by pressing the key combination: "Ctrl+alt+t" on you keyboard. You can also add the terminal shortcut to the favorites menu in your Ubuntu system, and open the terminal by clicking this shortcut instead. How to add the shortcut to you favorites menu is described in section 4.5 of this cookbook.

At this point we have a fully functional Ubuntu Virtual machine. In section 2.7 we are going to install the GeoStack Workshop software and content using the installation scripts provided in the Github Repository called: "GeoStack-Workshop". We are going to clone this repository later!

2.3 Connecting the Virtual Machine to the network

After you have started the GeoStack workshop virtual machine you should first check if the virtual machine has a network connection. This is done by performing the following steps:

1) First check if the network connection in the HOST Operating System of your computer.

2) Now check in your Guest OS of the Virtual Machine if your internet network connection is working by opening a terminal (ctrl + alt + t) and entering the command: ping 8.8.8.8. This command will send packets to the Google DNS server (8.8.8.8). This process will only work if you have an active network connection.

Let this command run for 3 seconds and then press ctrl + c to stop the pinging. The output should be similar to the one shown in the illustration below.

```
geostack@geostack-system:~$ ping 8.8.8.8
PING 8.8.8.8 (8.8.8.8) 56(84) bytes of data.
64 bytes from 8.8.8.8: icmp_seq=1 ttl=63 time=27.1 ms
64 bytes from 8.8.8.8: icmp_seq=3 ttl=63 time=34.2 ms
64 bytes from 8.8.8.8: icmp_seq=4 ttl=63 time=34.5 ms
^C
--- 8.8.8.8 ping statistics ---
4 packets transmitted, 3 received, 25% packet loss, time 3028ms
rtt min/avg/max/mdev = 27.104/31.985/34.562/3.453 ms
```

If the output is the same as the illustration below, we need to connect the Virtual machine to a network-connection.

```
geostack@geostack-system:~$ ping 8.8.8.8
connect: Network is unreachable
```

Connecting a Virtual Machine to a network is done by performing the following steps:

➔ Open the Virtual machine settings at the top of the screen.

➔ Go to the network settings and check the box: Enable Network Adapter and at "Attached to" select NAT (Network Address Translation) in the drop down box, as shown in the illustration below.
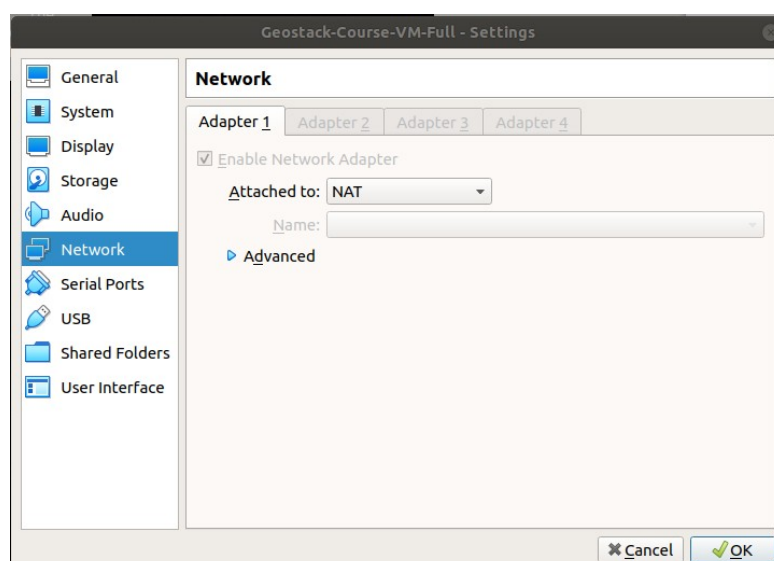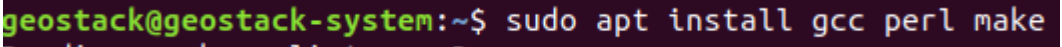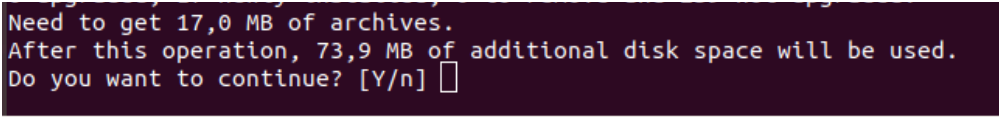

➔ Redo the previous steps to check if your internet connection is working. If it is you can move on to step 3. If it doesn't work you should consult the internet.

2.4 Creating a shared folder

A shared folder is used to transfer files from your host system to your Virtual Machine. To be able to create a shared folder we first need to install some missing software required for creating a shared folder between our host system and our virtual machine (guest system). These packages are GCC, Perl, Make and DKMS. There is a possibility that one or more of these packages are already installed on your system. We run the following steps to make sure these packages are installed and up to date. After installing these packages we need to install the Guest Additions ISO. Then we need to create the shared folder by linking a folder from our Host system to our Guest system. After the folder had been created, we need to grant this folder permissions to be able to access the folder.

2.4.1 Installing the required packages

We start of by installing some packages which are required to install the VirtualBox Guest Additions. This is done by performing the following steps:

➔ Open a Terminal windows by pressing the key combination: Ctrl + Alt + t on your keyboard.

➔ Then run the command: sudo apt install gcc perl make dkms

➔ Enter the password: "geostack", when prompted.

➔ Type y and press ENTER to let the install process finish. This process will take about 2 minutes to complete. If the packages are already installed you will see a message that the newest version is already installed.

➔ When the installation process is finished you can close the terminal by entering the command: exit.

2.4.2 Installing the VirtualBox Guest Additions

Now we are going to install the VirtualBox Guest Additions. The Guest Additions are required to create shared folders, enable full screen mode etc. For more information related to the VirtualBox Guest Additions you should visit the URL:
https://www.virtualbox.org/manual/ch04.html

The Oracle VM VirtualBox Guest Additions for all supported guest operating systems are provided as a single CD-ROM image file which is called VBoxGuestAdditions.iso. This image file is located in the installation directory of Oracle VM VirtualBox. To install the Guest Additions for a particular VM, you mount this ISO file in your VM as a virtual CD-ROM and install from there.

NOTE: It's possible that the Guest Additions ISO has already been mounted during the installation process of the Ubuntu VirtualBox. If this is the case you will get an error when performing step 1 in the steps on the next page.

This error is as follows:

Unable to insert the virtual optical disk /usr/share/virtualbox/VBoxGuestAdditions.iso into the machine GeoStack-Workshop-VM. Could not mount the media/drive '/usr/share/virtualbox/VBoxGuestAdditions.iso' (VERR_PDM_MEDIA_LOCKED).

To solve this you open File / Virtual Media Manager / Optical Discs / Click Guest Additions ISO and Click Release icon to detach the ISO from the VM.

So let's install the VirtualBox Guest Additions ISO file in our Virtual Machine by performing the following steps:

1) At the top of the VirtualBox window, click the Devices menu and then click the menu option 'Insert Guest Additions CD image.
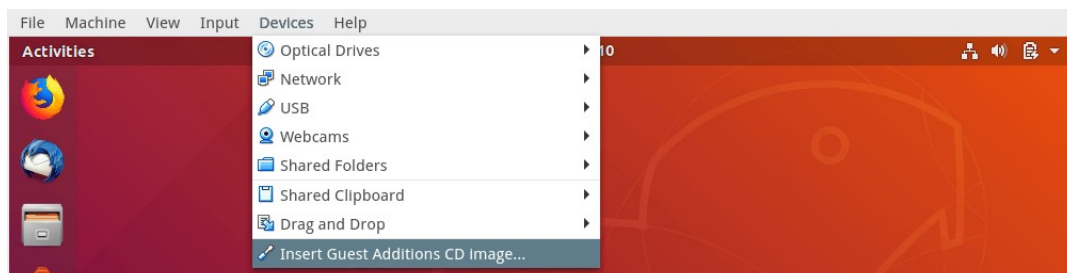
2) Click on run when the popup shows.
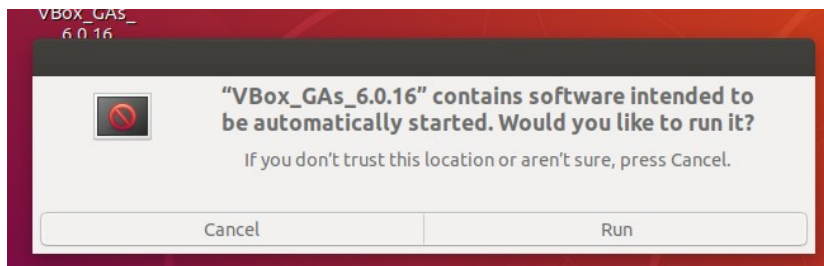
3) Enter the password: "geostack".
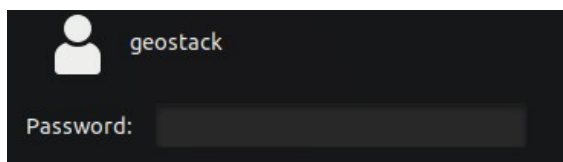
4) Wait till the install process is finished and close the terminal by clicking on the terminal window and pressing enter. The installation process takes about 1 minute to complete. Ignore the message in the terminal window about restarting to load the new kernel module that was compiled for the GA CD image because we will reboot the VM in the next step.
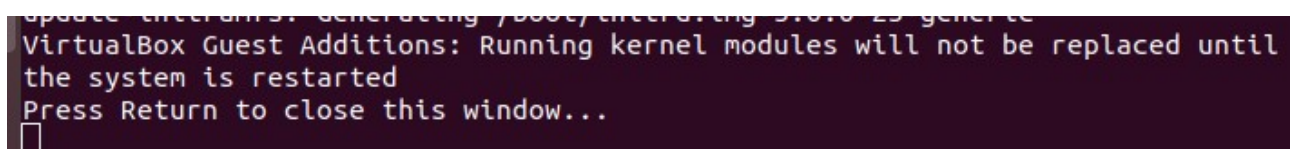
2.4.3 Adding a Shared Folder

Now let's add a shared folder by performing the following steps:

1) First create a folder on your **HOST** system. You can name the folder whatever you like but it's useful to give it a name related to the project you are currently working on. In this example we are going to name the folder: "TGP" which stands for The GeoStack Project.

2) After you have created the folder on your **HOST** system you should go to back to your Virtual Machine and select the following at the top of the Virtual Machine window: machine → settings → shared folders → press the Add button which will then open a popup screen.
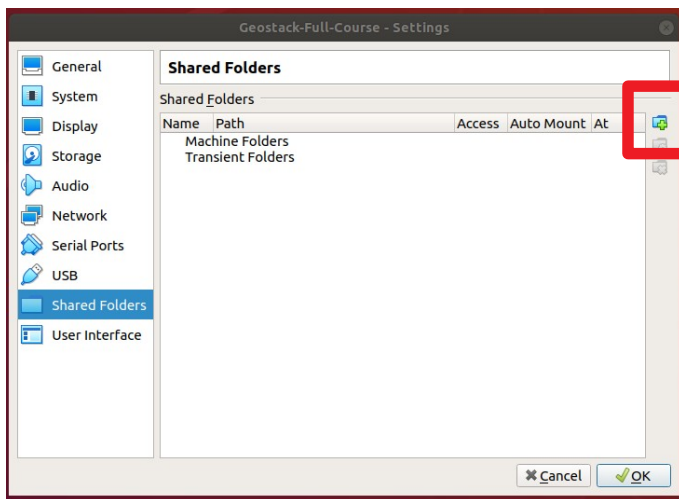
3) Select the path to where you created your folder (TGP in this case) on your host system as folder path:
   - ➔ The folder name should be automatically filled in.
   - ➔ Select auto mount.
   - ➔ Depending on you VirtualBox version, you should get the option: Mount point. You only get this option when you are running VirtualBox version 6.X. Set the mount point to: "Shared" in case you are using Vbox V6.
   - ➔ Select make permanent. This option automatically mounts the folder when the system is rebooted.
   - ➔ Press OK twice to close the windows.
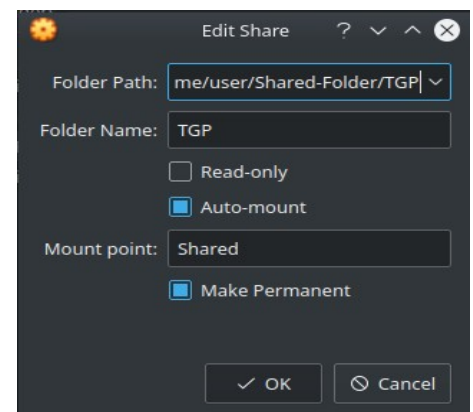
4) To be able to use the shared folder without having to enter a password every time we want to open the folder. We do this by opening a terminal by pressing the keys Ctrl + Alt + t (This is a keyboard shortcut for opening the terminal) on your keyboard and entering the command: sudo usermod -aG vboxsf geostack

The command: "usermod" is used to add the geostack user to the VirtualBox user group for the shared folders (the vboxsf group) . This is done by adding the flag -aG (-aG = add to Group) to the command.

5) To load the new usermod setting, reboot the virtual machine by entering the command: `reboot`
NOTE: You can also reboot the system by clicking the pull-down menu arrow at the upper right on the Ubuntu taskbar at the top of the Ubuntu desktop and there click the Power button and finally click Restart in the pop-up window.

6) Now the shared folder is available in the Ubuntu VM and can be accessed with the Nautilus file manager which can be started by clicking the file cabinet icon in the left Favorites menu. In Ubuntu 18.04 LTS the folder icon looks the same as shown in the illustration below.

7) When clicking on this icon a screen will open from which you can access the shared folder by clicking on the sf_TGP entry in the left pane of the Nautilus file manager as shown in the illustration below. Note: The sf_ stands for Shared folder.

8) Create a Desktop Shortcut for the Shared Folder in **Ubuntu 19.10 and 20.04** can be done by running the following command: `ln -s /media/sf_TGP ~/Desktop`

The shared folder is /media/sf_TGP which is mounted on a different partition (/media) than the home folder for accounts (/Home/accountname) so we don't get a 'Copy to...' menu option in the pop-up menu on a right mouse click on the shared folder in the Nautilus file manager. Therefore the desktop shortcut needs to be created from the terminal as a symbolic link (symlink).

Note: when placing a file in a shared folder from either the Guest OS or the Host OS first refresh the screen in the Nautilus file manager by pressing F5 or changing to another folder and then back if you don't see the file or if menu options like 'Move to trash...' are missing from the pop-up menu if you right-click a file with the mouse.

Note: also be aware that a shared folder does not have a Trash Can folder, so when you click on 'Move to trash...' for a selected file, this will result in a pop-up window with an error message 'Unable to find or create trash directory for /media/sf_TGP/filename.extension'. You can only permanently delete the selected file from the shared folder by clicking the red 'Delete' button in the pop-up window.

2.5 Installing the GeoStack Workshop software

In this section we are going to install all the tools, libraries and software required for the GeoStack Workshop. We are going to start of by cloning the Github repository which contains the GeoStack Workshop files to our Virtual Machine. This is done by performing the following steps:

1) Open a terminal by pressing the Ctrl + alt + t on your keyboard.

2) Install GIT by using the following command: sudo apt install git

3) Clone The GeoStack-Workshop Github repository by running the following command:

   git clone https://github.com/The-GeoStack-Project/GeoStack-Workshop.git

   The output of this command should be similar to the one shown in the illustration below:

   The download process takes about 10 seconds to complete and uses around 60MB of disk space. If everything works accordingly you should end up with a folder called: "GeoStack-Workshop" in your home directory of your virtual machine.
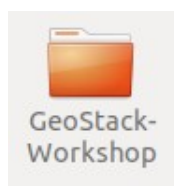
2.5.1 Downloading the Workshop datasets

As previously mentioned; you will be using 2 types of datasets during the GeoStack workshop. These are Crane (Tracker) Datasets and GPS Route (Trail) Datasets. The GeoStack Project is the owner of the GPS Route datasets so they are already provided in the GeoStack-Workshop folder which you downloaded in the previous chapter. The GeoStack-Workshop folder contains a sub-folder called: "GeoStack-Workshop-Content" which contains a sub-folder called: "Workshop-Datasets" in which you can find the GPS Route datasets. The CSV folder is empty since the GeoStack Project doesn't own the rights of the Crane (Tracker) Datasets. In this chapter we are going to download these datasets.

The datasets are provided by an organization called MoveBank. The Movebank Data Repository contains published data sets of animal movement data in the Movebank format. This is distinct from the main Movebank tracking database, in which users control access and are responsible for their data quality, and where most data are stored. To be published in the Movebank Data Repository, a data set in Movebank undergoes an official review process and, when accepted, is granted a unique identifier (DOI) and license and is made publicly available.

2.5.1.1 Downloading the Crane Tracker datasets

The way we are going to download the Crane (Tracker) datasets is by navigating to the Movebank website and downloading the complete datasets. This procedure is described in the steps below.

1) Navigate to the Movebank downloading page:
https://www.movebank.org/panel_embedded_movebank_webapp
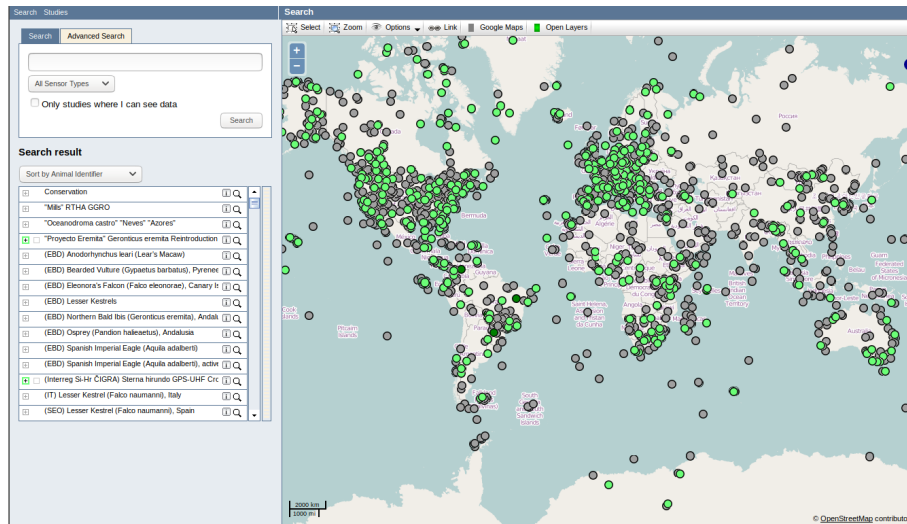
2) In the search field type: "grus grus" and click on search as shown in the illustration below:

3) Scroll down till you find the entry : GPS telemetry of Common Cranes, Sweden (n=19) as shown in the illustration below:

4) Click on the plus sign to expand the search results.

5) Select the box with the ID-number : 9381

6) Click on the information icon and then click on Download Search result:

7) Click on I agree in the screen that pops up:

8) Click on Download in the screen that pops up:

9) A Screen will pop up asking you to save the file. Select Save File and click on OK as shown in the illustration below:
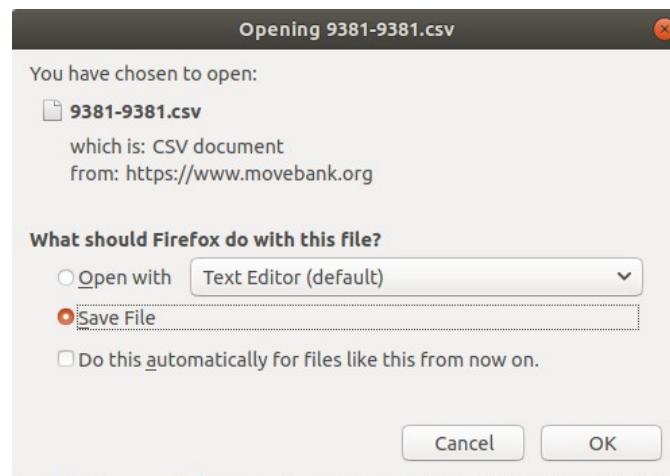
10) Now that you know how to download a Crane Tracker dataset you should repeat step 5, 6, 7, 8 and 9 on the Crane Tracker datasets with the id's 9407 and 9472 as shown in the illustration below:

11) After you have downloaded the 3 datasets you will end up with 3 CSV files in your download folder as shown in the illustration below:
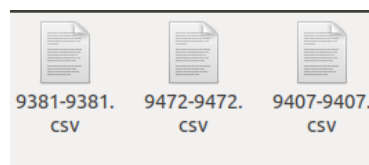
12) The last thing we need to do is to rename the Files to something logical and copy them to the correct place in our system. Currently the names of downloaded files represent the Tracker ID's of the Cranes. Since we want to know which Crane dataset is which, we are going to give the datasets a more descriptive name. For example: We are going to give the Crane Tracker with the ID: "9381" the name: "Frida". We also know that it's a Swedish Crane so we want to add that to the name. We also want the date from when the dataset was uploaded on the MoveBank website (10-03-2018). We also want to add the Ring color code to the name which is RRW_BuGBk(Red Red White - Blue Green Black). So the final name of the Crane (Tracker) Dataset with the Tracker ID: "9381" will be as follows: "20181003_Dataset_SV_GPS_Crane_9381_STAW_Crane_RRW-BuGBk_Frida.csv".

To rename and copy all the datasets to the correct place we should run following commands from a terminal:

For the Crane with the Tracker ID: "9381":

```
cp ~/Downloads/9381-9381.csv
~/GeoStack-Workshop/Geostack-Workshop-Content/Workshop-Datasets/CSV/
20181003_Dataset_SV_GPS_Crane_9381_STAW_Crane_RRW-BuGBk_Frida.csv
```
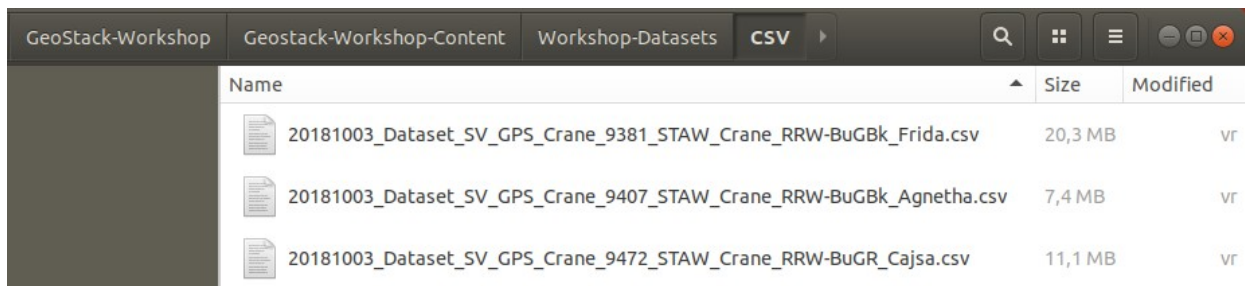
For the Crane with the Tracker ID: "9407":

```
cp ~/Downloads/9407-9407.csv
~/GeoStack-Workshop/Geostack-Workshop-Content/Workshop-Datasets/CSV/
20181003_Dataset_SV_GPS_Crane_9407_STAW_Crane_RRW-BuGBk_Agnetha.csv
```

For the Crane with the Tracker ID: "9472":

```
cp ~/Downloads/9472-9472.csv
~/GeoStack-Workshop/Geostack-Workshop-Content/Workshop-Datasets/CSV/
20181003_Dataset_SV_GPS_Crane_9472_STAW_Crane6_RRW-BuGR_Cajsa.csv
```

If everything is done accordingly you should end up with the following datasets in the folder "GeoStack-Workshop/GeoStack-Workshop-Content/Workshop-Datasets/CSV"

2.5.2 Pre-installation Information

Now that we have the necessary files and datasets on our Virtual Machine we can start the installation process. First we need to clear some things up.

NOTE: INSTALLING THE SOFTWARE AUTOMATICALLY, USING THE INSTALLATION SCRIPTS, DOES NOT MEAN YOU DON'T HAVE TO PAY ATTENTION.

Once in a while you are asked for some input such as changing default passwords, accepting updates and adding package sources to our system's sources list. The following should be done in those cases:

1. **Password prompt enter the password: geostack**

2. **Accepting updates by entering: y**

3. **Adding packages to the source list of software repositories: press ENTER**

Now that is cleared up we can start installing the software. The folder: "Installation-scripts", contains 8 scripts. The backend, middleware and frontend software each has it's own installation script.

2.5.3 Running the Workshop Installation Scripts

Now let's start with the execution of the installation scripts:

1) Close all windows and open a new terminal by pressing the keys Ctrl + Alt + t on your keyboard.

2) Then run the first script by entering the command:
   `bash ~/GeoStack-Workshop/Installation-scripts/1-pre-reboot.sh`

   This script will do the following:

   ➔ Update the system.

   ➔ Install open-vm-tools.

   ➔ Install Bleachbit.

   ➔ Install net-tools.

   ➔ Install Python3 and Python3-pip.

   ➔ Install Atom.

   ➔ Add the sidebar shortcuts which we are going to need during the workshop.

   After the completion of this script the system will reboot for the updates to take effect. This script takes around 10 minutes to complete depending on the speed of you network-connection. After the script is complete 2GB of additional disk space is used.

3) After the reboot open the terminal again and enter the command:

   `bash ~/GeoStack-Workshop/Installation-scripts/2-post-reboot.sh`

   This script will do the following:

   ➔ Copy the desktop shortcuts to the correct place.

   ➔ Create file links on the desktop so you can easily access the workshop folders.

   This script will make sure all the folders and shortcuts are placed in the correct place and takes about 2 minutes to complete. After the script is complete no additional disk space is used.

4) Now when that script is done and no errors were encountered we can move on to the script, for installing the data-analyses software required for performing the data analyses in part 1 of the workshop, by running the command:

   `bash ~/GeoStack-Workshop/Installation-scripts/3-data-analysis-software.sh`

   This script will do the following:

   ➔ Install Jupyter Lab

   ➔ Install Pandas

   ➔ Install Cartopy and MathPlotLib

   ➔ Install GPXPy

   This scripts takes about 10 minutes to complete depending on the speed of your network-connection. After the script is complete 1.4GB of additional disk space is used.

5) Now when that script is done and no errors were encountered we can move on to the next script, for installing the backend software required to store all the data in part 2 of the workshop, by running the command:

bash ~/GeoStack-Workshop/Installation-scripts/4-backend-software.sh

This script will do the following:

➔ Install MongoDB.

➔ Install MongoCompass.

➔ Add the new sidebar shortcuts.

➔ Install MongoEngine.

➔ Clean unnecessary files.

This scripts takes about 10 minutes to complete depending on the speed of your network-connection. After the script is complete 1GB of additional disk space is used.

6) Now when that script is done and no errors were encountered we can move on to the next script, for importing all the datasets in the correct datastores, by running the command: bash ~/GeoStack-Workshop/Installation-scripts/5-dataset-import.sh

This script will do the following:

➔ Import the Crane (Tracker) datasets in MongoDB.

➔ Import the GPS-Route (Trail) datasets in MongoDB.

➔ Copy the workshop Part 1 datasets to the correct location on the system.

➔ Copy the workshop Part 2 datasets to the correct location on the system.

This script is used to import all the data which can be used in case something goes wrong during part 2 of the workshop. The script takes about 4 minutes to complete and an additional disk space of 200MB is used.

NOTE: During the import process the VM can become slow. Don't worry about this! The reason for this is because the MongoDB import scripts use a feature called: "bulk import". This feature adds all the data at once. This makes the import process a lot faster but can take up a lot of RAM which results in a slower VM (temporarily).

7) Now when that script is done and no errors were encountered we can move on to the next script , for installing the middleware software required for part 3 of the workshop, by running the command:

bash ~/GeoStack-Workshop/Installation-scripts/6-middleware-software.sh

This script will do the following:

➔ Install Python-Flask.

➔ Install Flask-Pymongo.

➔ Install Gunicorn3.

➔ Install TileStache and Pillow.

➔ Install NGINX.

This scripts takes about 10 minutes to complete depending on the speed of your network-connection. After the script is complete 1.2GB of additional disk space is used.

8) Now when that script is done and no errors were encountered we can move on to the next script , for installing the frontend software required for part 4 of the workshop, by running the command:

`bash ~/GeoStack-Workshop/Installation-scripts/7-frontend-software.sh`

This script will do the following:
➔ Install NodeJS.
➔ Install the Angular CLI.
➔ Install the Node Modules for the web applications.

NOTE: During the execution of this script you are asked to share anonymous usage data with the Angular Team at Google under Google's Privacy Policy. Select No and press enter in all cases. During the script you will also see some NPM warnings. Don't worry about these warnings. These security / update warnings are normal when installing Angular Node modules. If you want to find more information related to updating Angular modules, you should follow the complete GeoStack course.

This scripts takes about 3 minutes to complete depending on the speed of your network-connection. After the script is complete 300MB of additional disk space is used.

9) Now when that script is done and no errors were encountered we can move on to the last script which removes all the unused packages and clears the temporary files. We do this by running the command:

`bash ~/GeoStack-Workshop/Installation-scripts/8-post-installation-cleanup.sh`

This script takes about 3 minutes to complete and clears about 700MB of disk space. After this script is complete the total size of the Virtual disk will be around 8.7 GB

10) If you are on **Ubuntu 19.10 or 20.04** you need to allow the desktops shortcuts to be executable. This is done by right-clicking a desktop shortcut and the selecting Allow Launch in the menu that pops up as shown in the illustration below:
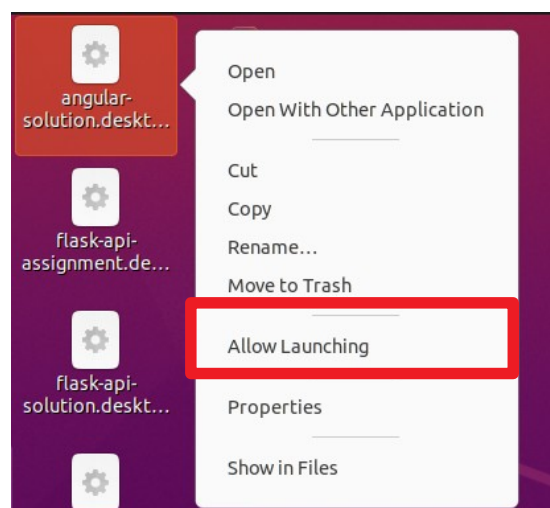
11) After the last script is finished running you will end up with a lot of shortcuts on your desktop. You should organize the desktop icons in the same way as shown in the illustration below. This is done by dragging and dropping the shortcuts in the correct position and will result in a better workflow. **Note: This screenshot was created using Ubuntu 20.04.**
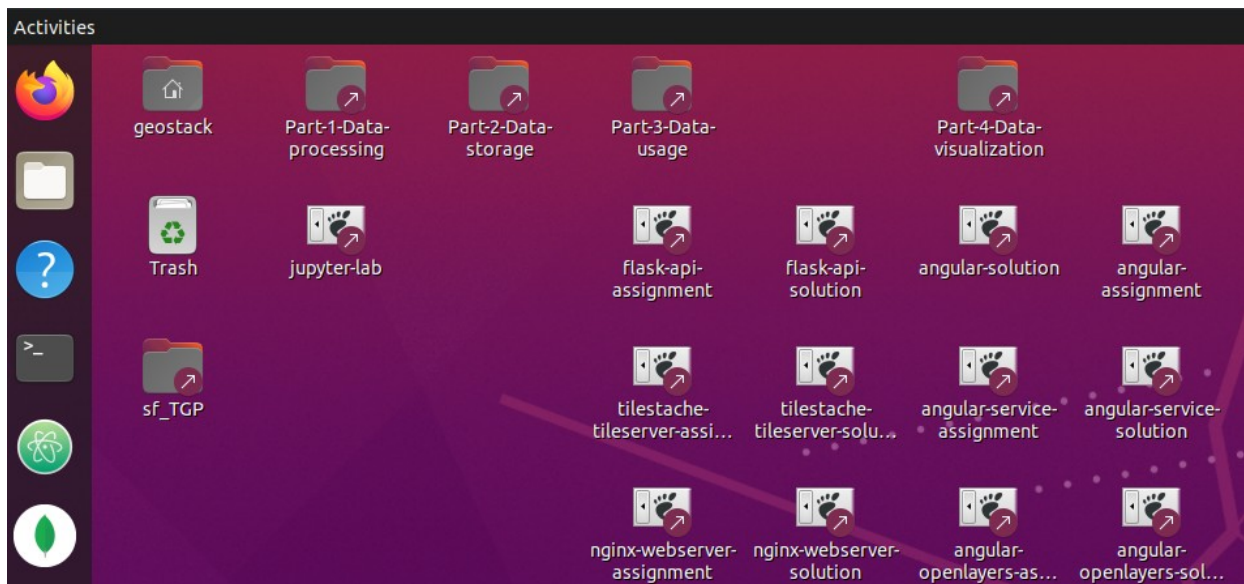
That's it ! When the last script is finished you can start the Workshop by reading the PDF called: "Workshop-Assignments.pdf". This document contains all the instructions for the assignments which you have to complete during the GeoStack Workshop. The document can be found in the root folder of the GeoStack-Workshop folder which you downloaded at the beginning of chapter 2.5

3. Managing Virtual Machines

The easiest way to get a Virtual Machine up and running is by using .OVA files. The easiest way to backup and distribute your Virtual Machines as one (1) file for easy portability is to export your Virtual Machine with its Virtual Disk Image to what is called an OVA file (Open Virtualization Archive).

An OVA file is an Open Standard in the Open Virtualization Format (OVF) to package an entire VM into a single file. It can be used by virtualization applications such as Vmware Workstation and Oracle VM Virtualbox.

You also have the option to export a separate human readable Manifest companion file for the OVA file with its description and some meta data.

You can also include the ISO image files of the Operating System and the Guest Additions into the OVA file for easy backup or to let another person reuse the same ISO images.

Creating an OVA file is also done as a way to make a snapshot of a VM to freeze it's contents to preserve a state of installation or use.

NOTE: Make sure you have enough disk space!

It's also possible to import and export .ova files in VMWare. The instructions on how to do this can be found by clicking on the following URL:

https://pubs.vmware.com/fusion-5/index.jsp?topic=%2Fcom.vmware.fusion.help.doc%2FGUID-275EF202-CF74-43BF-A9E9-351488E16030.html

Tip: For working in development teams think about the password of the Linux account of the Virtual Machine. If you distribute your Virtual Machine as an OVA file to someone else then depending on your security requirements either hand over the password of the user account of the VM in a secure way or if security is not an issue put the password in a ReadMe.txt file that you distribute with the OVA file or even make it very easy by making the password the same as the account name!
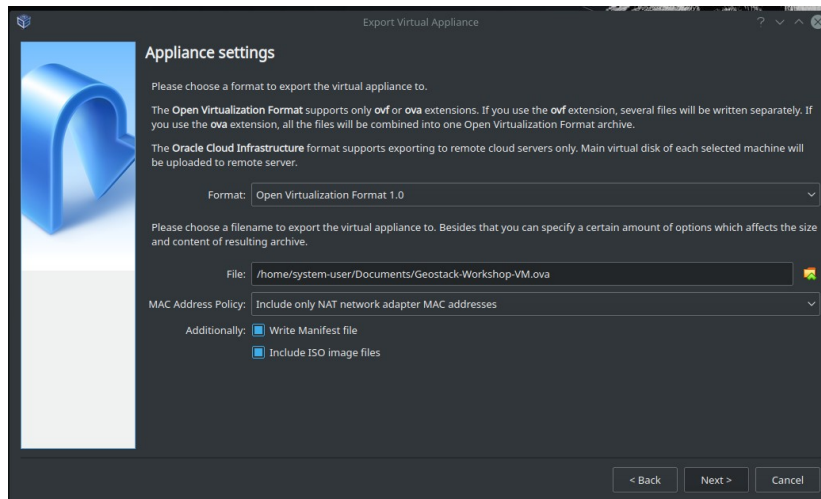
3.1 Exporting a Virtual Machine to an OVA file

To export a Virtual Machine to a .OVA file you have to perform the following steps:

1) In the VirtualBox window click on file → preferences → export appliance.

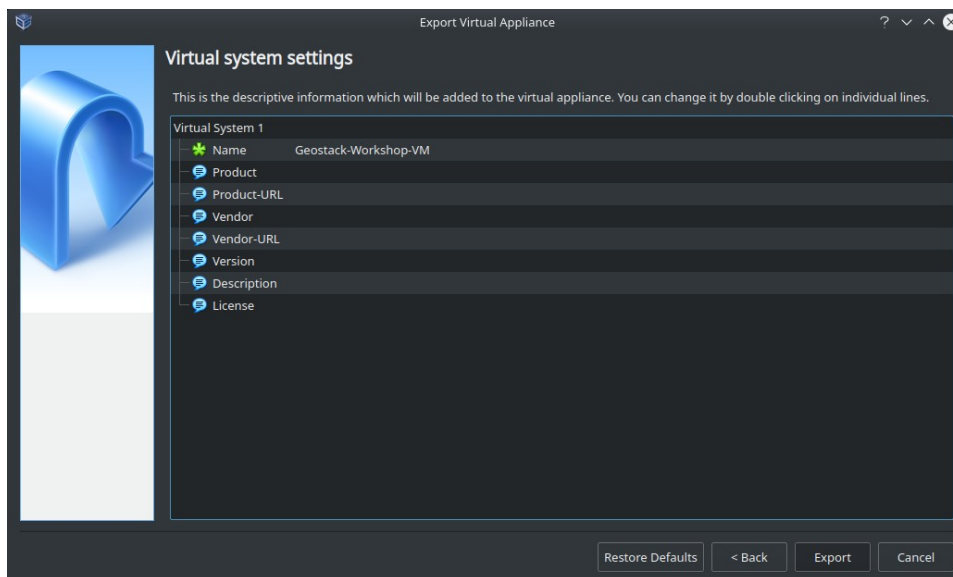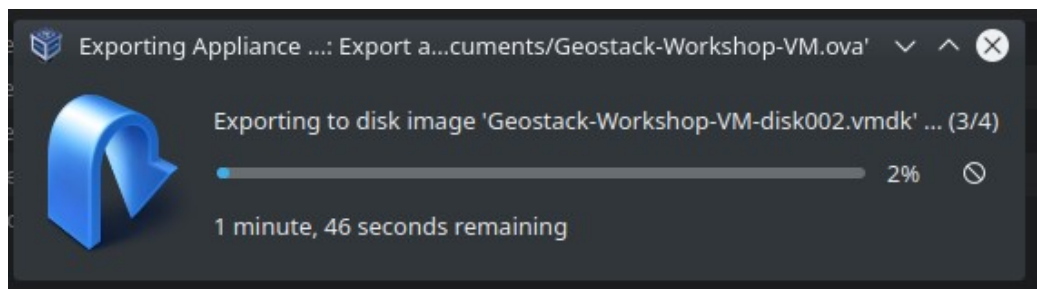2) Select the Virtual Machine which you want to export. In this case it's the GeoStack workshop Virtual Machine.
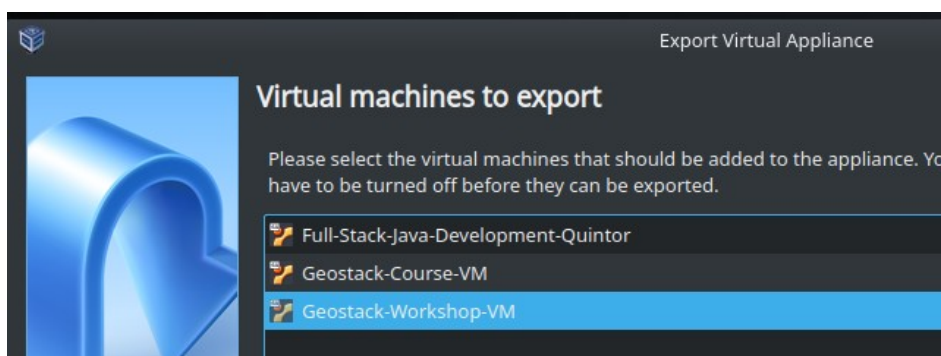
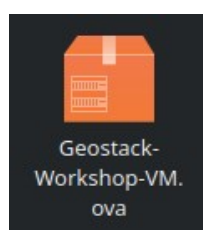3) Set the settings to be the same as in the illustration below and click on next.

4) Check if the Name of the Virtual Machine is correct and click on export.

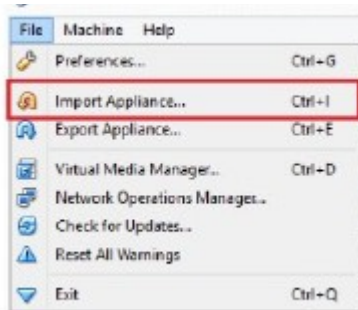5) Let the export process finish. This process takes about 10 minutes.

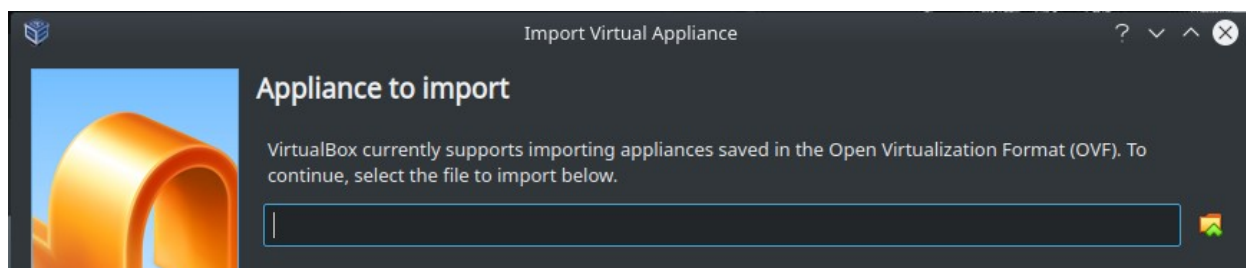6) Now you will have an OVA file which can be easily distributed.

3.2 Importing an OVA file

To import a Virtual Machine using a .OVA file you have to perform the following steps:
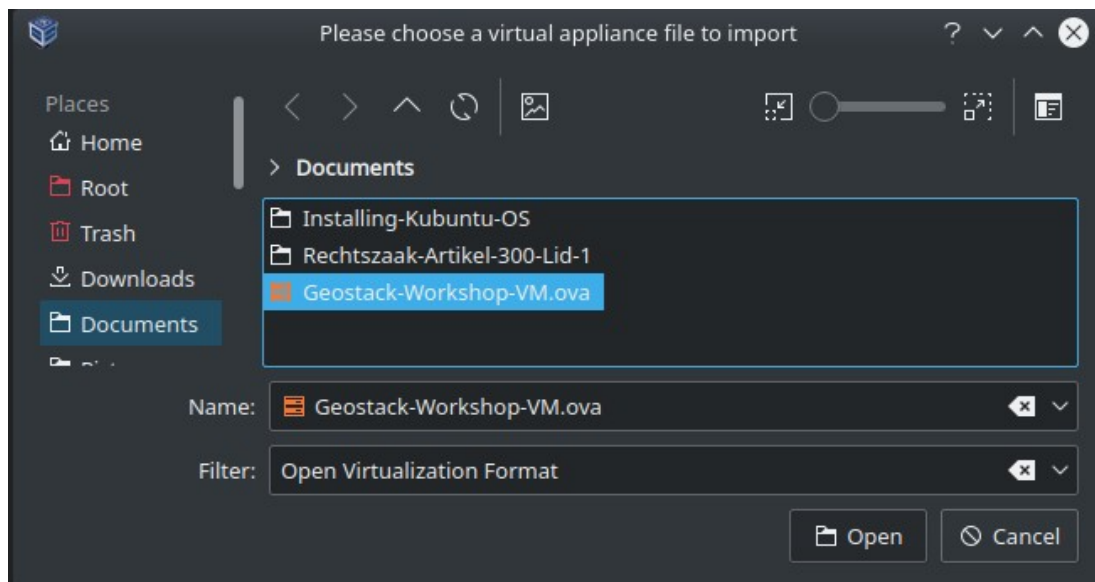
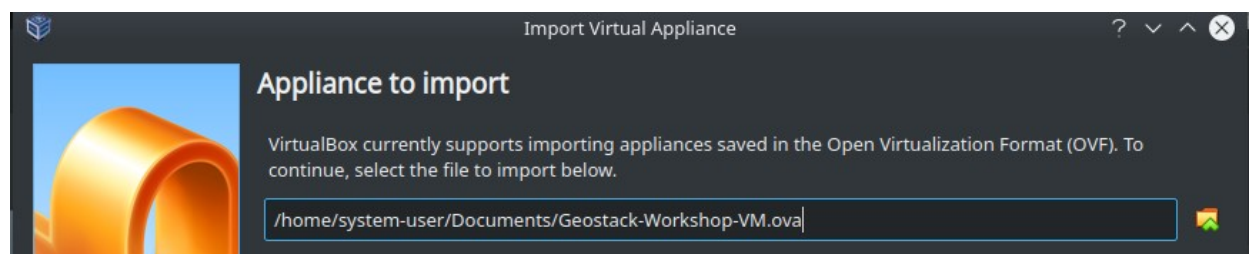1) In the VirtualBox window click on file → preferences → import appliance.
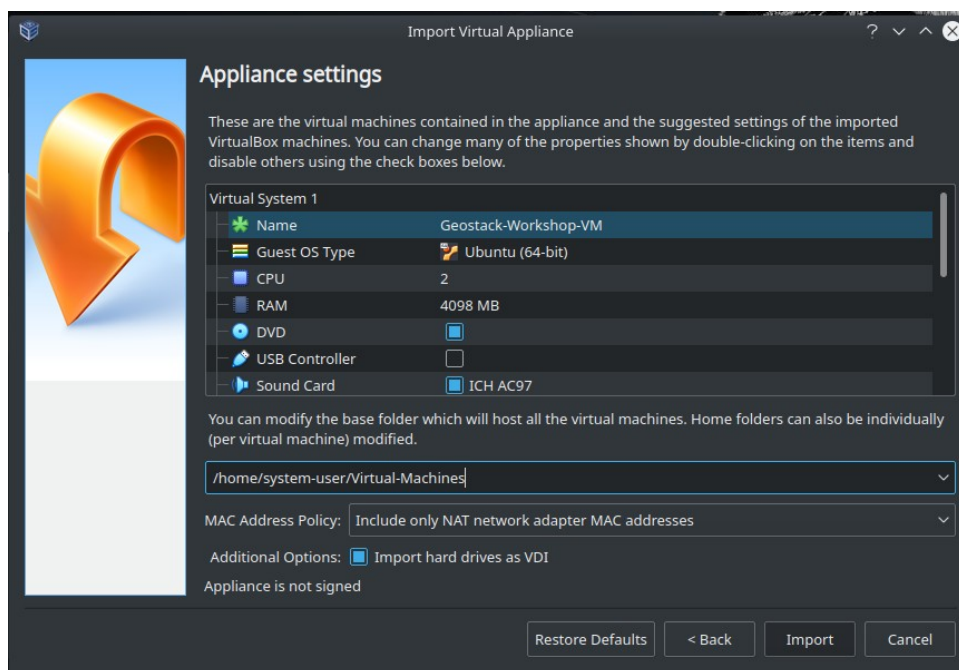

2) Click on the folder icon.

3) Navigate to the folder in which you saved the .ova file which was exported in the previous section and click on open.
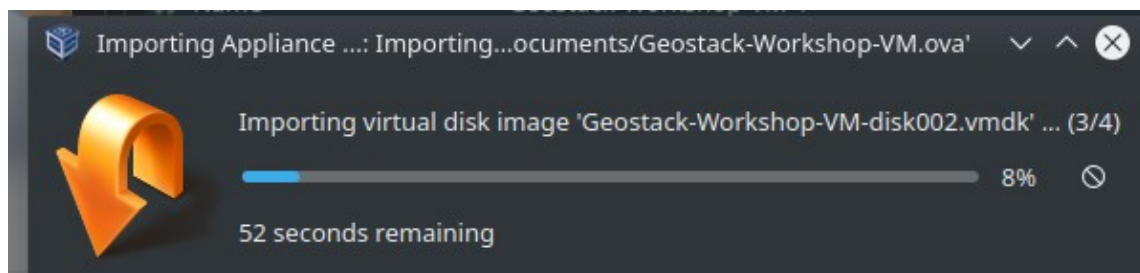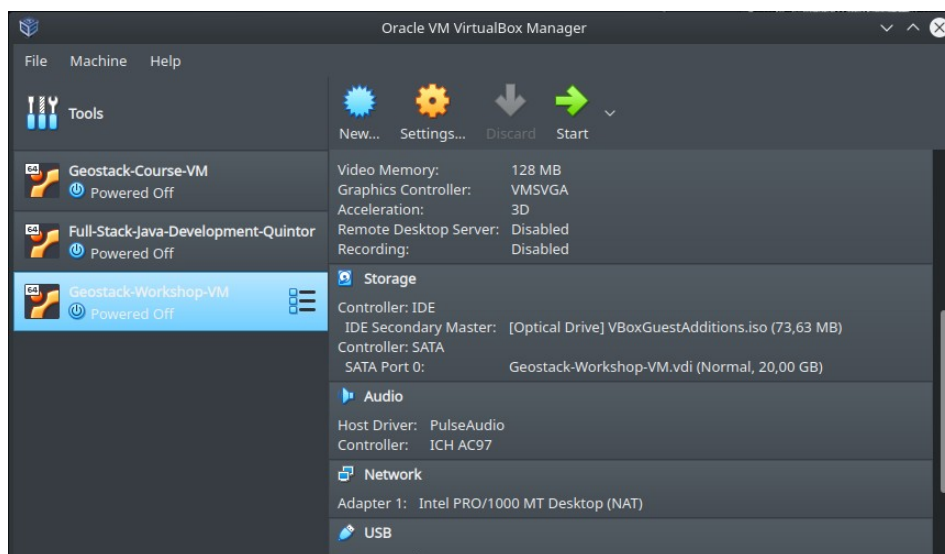

4) Click on next.

5) Change the settings where you find necessary and click on import.

6) Wait till the import process is finished. This will take about 3 minutes to complete.

7) That's it, now you have the full Geostack as Virtual Machine on you system. You can start the Virtual Machine by selecting it and pressing the green arrow at the top of the VirtualBox screen.

3.3 Increasing available RAM for the Virtual Machine

If you have decent laptop you can always increase the amount of RAM that the Virtual Machine can use. By doing this the Virtual Machine will work faster en smoother.

To increase the amount of RAM that can be used by the Virtual Machine you have to perform the following steps:

1) Shut down your Virtual Machine.
2) In the VirtualBox start screen go to settings → system and increase the amount of RAM used by the VM.
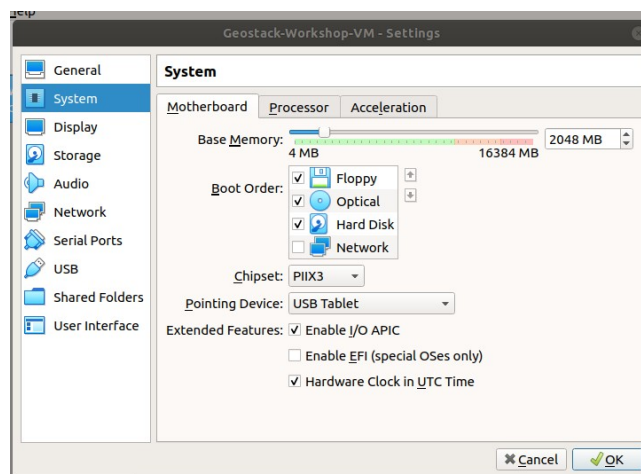
3.3 Increasing the Video Memory for the Virtual Machine

Setting the Video Memory to the maximum amount of 128 MB will ensure smooth screen handling by VirtualBox and prevent black screens or slow screen redraws when resizing the window of the Virtual Machine. To fix this you need to perform the following steps:

1) Shut down your Virtual Machine.

2) In the VirtualBox start screen go to settings → display.

3) Increase the Video Memory and toggle the Enable 3D and 2D acceleration boxes on.

You can also check the acceleration check boxes. For Linux as Host OS only check 'Enable 3D Acceleration'. For Windows as Host OS also check 'Enable 2D Video Acceleration'. If checked under Linux this 2D option will give the warning 'Invalid settings detected' as shown in the screen image below.
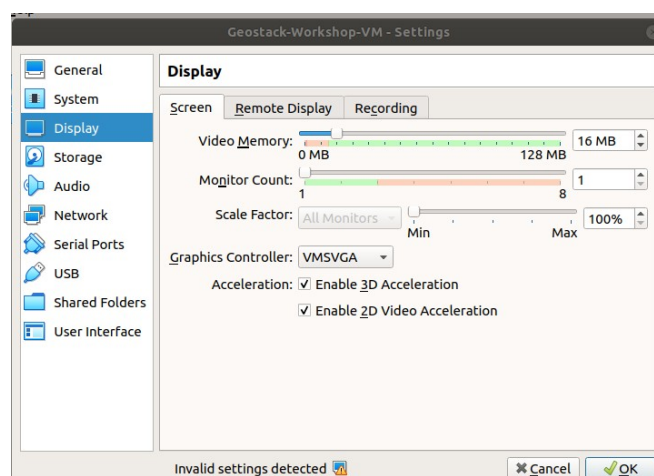
4. Useful tips and tricks

In this chapter some useful tips and tricks are discussed. These tips and tricks can increase the workflow during and after the workshop.

4.1 Disabling the Ubuntu lock screen

Removing the Ubuntu lock screen can come in handy since you don't have to login every 5 minutes after removing the lock screen. Removing the lock screen can be done by performing the following steps:

1) Open the show application menu on the bottom left of the desktop. The icon is shown in the illustration below.
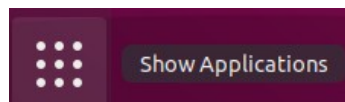
2) Enter: "Power" in the search box and select the setting shown in the illustration below.
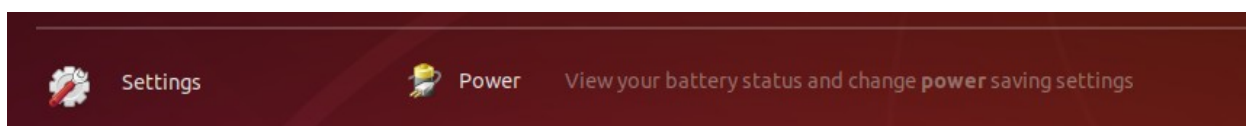
3) Then select the drop down box next to: "Blank screen" and set it to never as shown in the illustration below.

4.2 Manually updating Ubuntu using the CLI

To manually update the Ubuntu system using bash commands. you should perform the following steps:

1) Open a terminal by pressing the key combination: Ctrl + alt + t on you keyboard.

2) Retrieve the new packages and package updates by running the command: `sudo apt update`

3) Update the system using the command: `sudo apt upgrade`

4) Reboot the system for the updates to take effect by using the command: `reboot`

4.3 Manually updating Ubuntu using the GUI

To manually update the Ubuntu system using the Gnome GUI. you should perform the following steps:

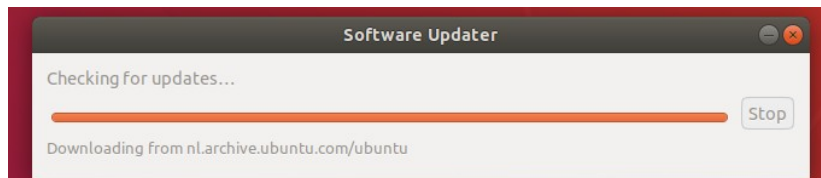1) Open the show application menu on the bottom left of the desktop. The icon is shown in the illustration below.
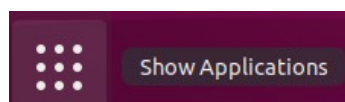
2) Then select All at the bottom of the screen that pops up, as shown in the illustration below.

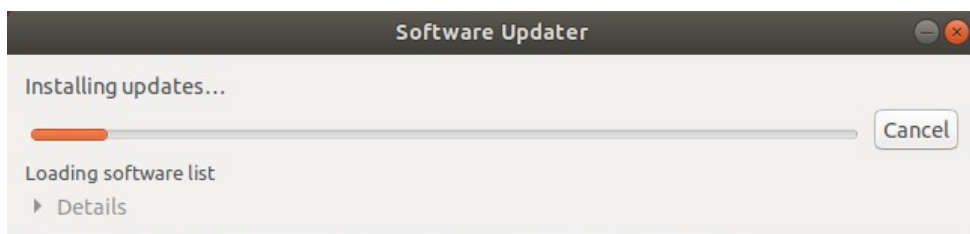3) Select the Software Update icon, as shown in the illustration below.
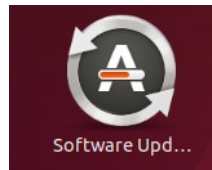
4) Let the update search for updates as shown in the illustration below.

5) Select: "Install Now" in the screen shown in the illustration below.

6) Let the software updates finish as shown in the illustration below.

7) When the screen in the illustration below is shown, the updates are successfully installed.

4.4 Cleaning the system

If you want clean your system after installing updates, you can perform the following steps:

1) Clean the unnecessary packages by running the command: sudo apt autoremove

2) Clean the system's memory and temporary files, using an application called: "Bleachbit", by running the following command:

   bleachbit --list | grep -E "[a-z0-9_\-]+\.[a-z0-9_\-]+" | grep -v system.free_disk_space | xargs sudo bleachbit --clean

   This command first prints the available cleaning options, then it greps every option except for the free disk space option since this is unnecessary. Finally Bleachbit is executed using the selected options.

4.5 Adding shortcuts to the Favorites menu

During the installation scripts some shortcuts are added to the Favorites Menu on your Ubuntu system. If you want to do this manually you have to perform the following steps:

1) Open the show application menu on the bottom left of the desktop. The icon is shown in the illustration below.
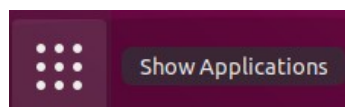
2) Then select All at the bottom of the screen that pops up, as shown in the illustration below.
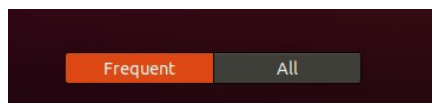
3) Enter the name of the application, which you want to add to the menu, in the search bar as shown in the illustration below. In this example we are going to add the terminal to our Favorites menu.
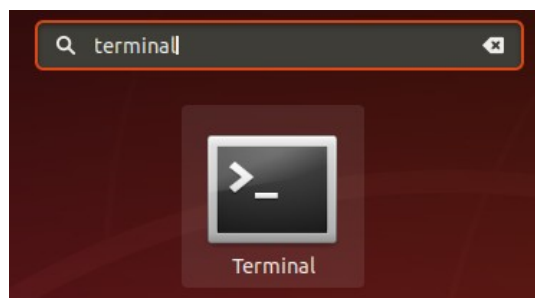

4) Right click on the terminal icon and select: "Add to Favorites" as shown in the illustration below.
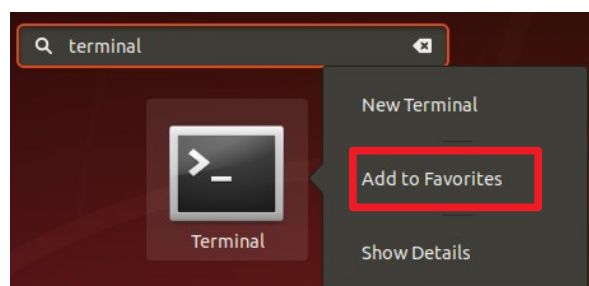
4.6 Clipboard sharing between Host and Guest

If you want to copy/paste text between your Host and Guest system, you need to activate this option by going to Devices → Shared Clipboard at the top of the VirtualBox Window and selecting: "Bidirectional" as shown in the illustration below.
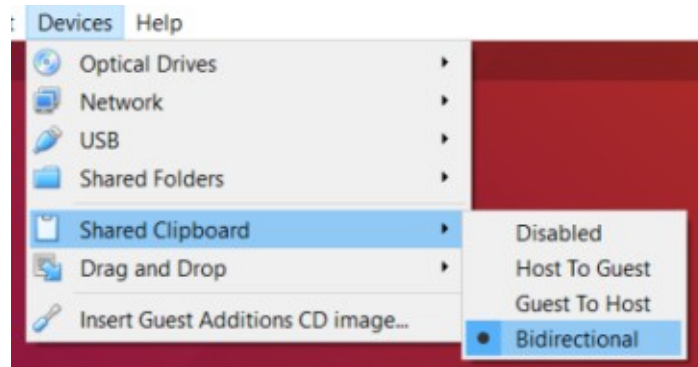
4.7 Drag and Drop between Host and Guest

If you want to drag and drop files between your Host and Guest system, you need to activate this option by going to Devices → Drag and Drop at the top of the VirtualBox Window and selecting: "Bidirectional" as shown in the illustration below.
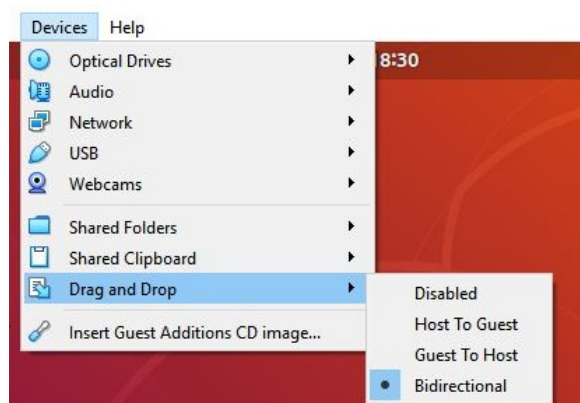
4.8 Installing LibreOffice writer

Because you installed the minimal version of the Ubuntu Linux distribution the office suite LibreOffice is not installed and it is required if you want to use the workshop's text editor and presentation files in their original file format to read or edit for your own purposes because that's why we included these source files along with their 'frozen content' versions in the PDF file format that you would normally use to follow the workshop.

The original source files are the ODT files (Open Document Text) from the text editor application LibreOffice Writer and the ODP files (Open Document Presentation) from the presentation application LibreOffice Impress.

To install the packages for the LibreOffice applications and help texts (this requires about 1 GB of disk space in total), open the Terminal application with the keyboard shortcut Ctrl + Alt + t and

then use the following commands to install the basic version of LibreOffice (about 915 MB in disk size): `sudo apt install libreoffice`

To install the package with English help texts (about 32 MB in disk size) use either the command: `sudo apt install libreoffice-help-en-us` (for help texts in English - American spelling)
or
`sudo apt install libreoffice-help-en-gb` (for help texts in Englis - Brittish spelling)

See the following weblink with instructions on how to install help text packages in other languages or other additional LibreOffice packages from the command line if you might need them:
https://wiki.ubuntu.com/LibreOffice

After the installation you will find the LibreOffice applications in the Show Applications menu (the nine-grey-squares icon at the bottom the Favourites menu at the left of the desktop screen) or double click an ODT or ODP file to open the corresponding Writer or Impress application respectively.