Nerve Blue 2.0 Documentation

Documentation Overview

This documentation will guide you through the Nerve Blue product.
It offers six guides, including a quick start guide to help you get operational. Please refer to the table below for an overview of each available guide. The unreleased guides will be added in the near future. Thank you for your patience.

Guide	Description
Quick Start Guide	If you want to get started and operational quickly, this is the guide for you. Please be aware that this guide only covers parts and does not give full details.
Concepts Guide *(not yet released)*	This guide introduces the concepts and components behind Nerve.
User Guide	This is the complete guide covering all relevant information for regular users.
Device Guide	This guide is dedicated to supported Nerve Devices and their functionality.
Developer Guide *(not yet released)*	Developers will find an introduction to possibilities of development here.
Nerve Blue Kit	A guide covering the usage of the Nerve Blue Kit.

Resources and Links

In the process of the set up and installation, you will need certain files to be able to complete the set up. Your sales representative or support has sent you all the files as part of the delivery.

The delivery also contains the customer profile that has information such as your credentials for the local UI and the Management System. Please keep the customer profile close by when reading through this guide.

If you have not received any files or a customer profile, please contact your sales representative or TTTech Industrial customer support at support@tttech-industrial.com.

Manufacturer Address and Support Contact

If you should have any questions about the software installed and the features provided, please contact your sales representative or TTTech Industrial customer support at support@tttech-industrial.com.

TTTech Industrial Automation AG
Schönbrunner Strasse 7
1040 Vienna Tel: +43 1 585 34 34-0
e-mail: support@tttech-industrial.com

Quick Start Guide

Quick Start Guide

This quick start guide will walk you through the basic setup of your Nerve Device and the usage of the Management System. At the end of this document you will be able to:

- Set up the Nerve Device
- Connect to the local UI of the Nerve Device
- Connect to the Management System in the cloud
- Provision CODESYS, Virtual machine and Docker workloads
- Deploy workloads to get ready and operational

This guide focuses on getting you up and running with Nerve Blue as fast as possible. It leaves more detailed information to following guides. It will not go into the specifics of your Nerve Device and uses the MFN 100 as an example. To find out specific information on your Nerve Device, please see the Device Guide.

Hardware Overview and Setup

This chapter will give you basic hardware information about the MFN 100 and show you how to install and set it up.
The MFN 100 is a qualified Nerve Device that is optimized and tested for use with Nerve software. The device is designed for use in harsh industrial environments.

Contents

The MFN 100 is shipped with a User an Installation Guide and a mating connector.

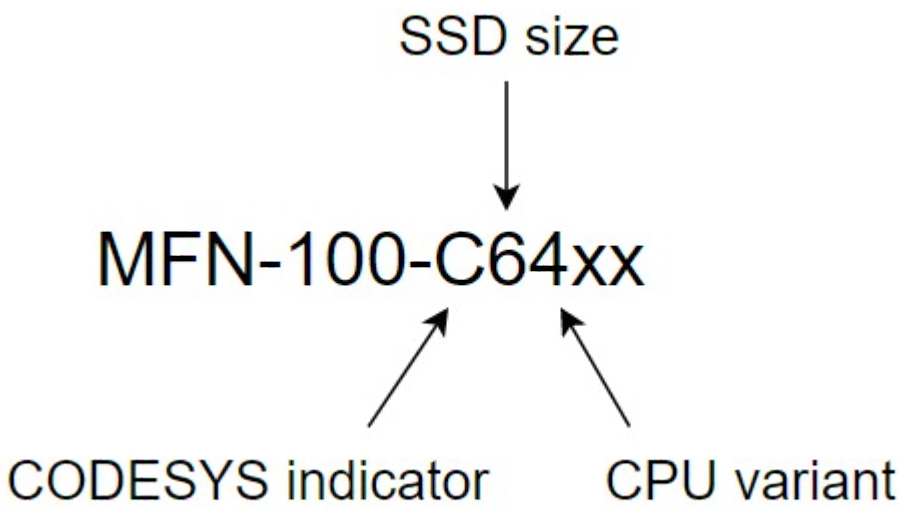
Hardware	Software
• MFN 100-<modelnumber> • Mating connector	**Pre-configured** • Management system hosted in Azure Cloud **Additional files** • MFN 100 device description file • Node registration tool

NOTE

You will also need to download the CODESYS Development System. Please visit store.codesys.com to do so.

Identifying the MFN 100

The label of the MFN 100 can be found on the back of the device, close to the DIN rail clip. Exact identification is possible through the combination of product number (P/N), serial number (S/N) and version number (V/N) that are printed on the label. The model number of the MFN 100 details the variant of the MFN 100 you have purchased:



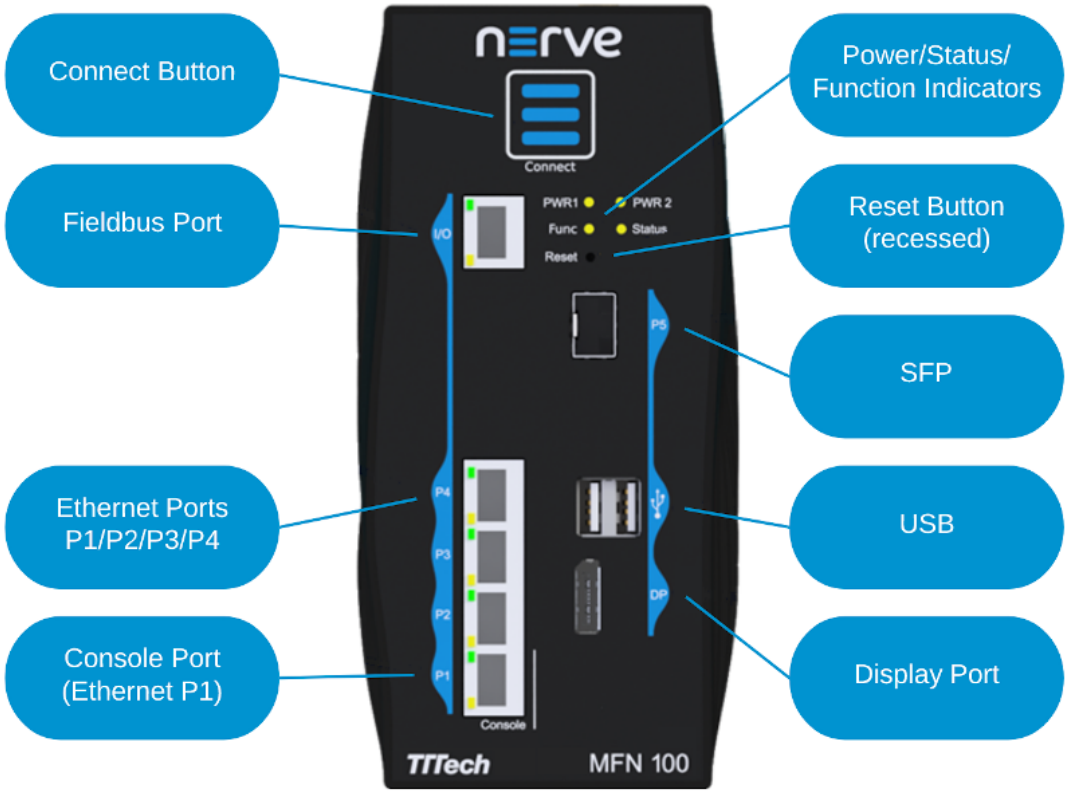
Letter or Number	Description
CODESYS indicator	This letter indicates whether the device has a CODESYS runtime pre-configured: • **C** — The CODESYS runtime is pre-configured • **X** — The CODESYS runtime is not pre-configured
SSD size	This number indicates the size of the SSD: • **6** — 64 GB SSD • **2** — 256 GB SSD • **5** — 512 GB SSD
CPU variant	This indicates the CPU variant of the device: • **4** — Intel E3940 (4 GB RAM) • **5** — Intel E3950 (8 GB RAM)

NOTE

Please make sure to write down the serial number of your Nerve Device. You are going to need it for the node registration process.

Front Panel Controls and Indicators

Below is an overview of the front panel of the MFN 100, describing physical interfaces, indicators and their labels.

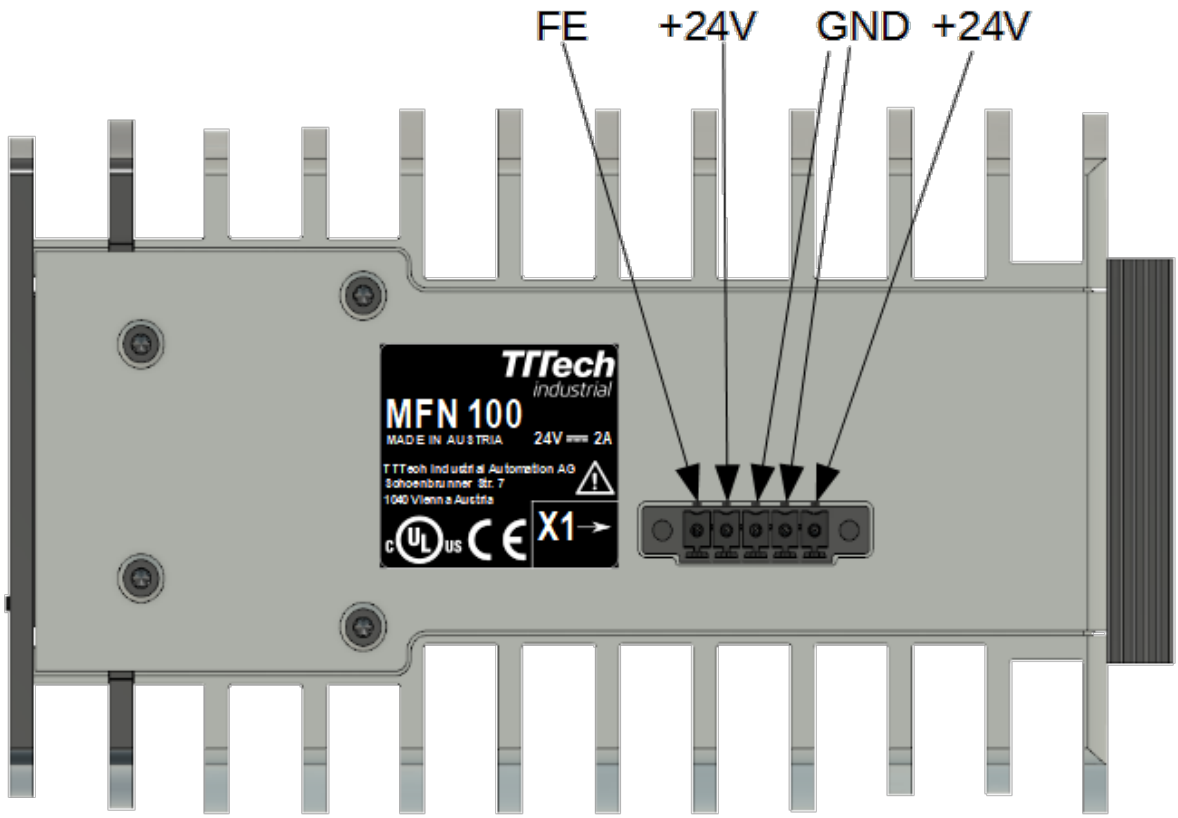


Label	Description
Connect Button	The connect button interrupts the connection on ports P2 to P5 of the MFN 100. This is the behavior in the standard configuration. The function is configurable on request. The button may be configured to change the network configuration.
Connection Indicator	The connection indicator is the first fin in the MFN 100 housing. It lights up blue when all required services are initiated and the connection to the Management System is configured.
Reset	Holding the button for 4-8 seconds initiates a power cycle. Use a tool with a rounded tip to press the button.
Power 1 **Power 2**	Indicators showing power active on the power supply inputs.
Status	LED indicating system status • Green: All device functions are ready. • Not lit: Device functions are not ready or the device is booting.

Label	Description
Function	LED indicating CODESYS runtime status • Green: CODESYS runtime is operational. • Not lit: CODESYS runtime is not operational.
P1 Console	Ethernet port/console port. This port is typically used to connect a workstation to configure the MFN 100.
P2/P3/P4	Ethernet ports
P5	SFP port
I/O	Fieldbus interface
USB	Two USB 2.0 ports with 1.1 A maximum output current for both ports combined.
DP	Display Port supporting the DP++ standard.

Power Connectors Overview

The power connectors are located at the bottom of the MFN 100 next to the label. There are two separate 24 V inputs, two GND inputs and one Functional Earth (FE) input. The inputs are fused internally. The fuse cannot be replaced by the user. The power supply inputs are protected against reverse polarity.



Pin	Description
1	Functional Earth (FE)
2	Power supply line 1
3	GND
4	GND
5	Power supply line 2

NOTE

The GND and FE pins (pins 2, 3, and 5) are electrically connected to the housing.

Power Supply Details

Parameter	Value
Operating voltage	18 - 30 VDC
Start-up current	7 A max.
Consumption	1.4 A continuous 2.1 A peak
Dissipated power	33.6 W at 24 VDC

Installation and Removal on a DIN Rail

The MFN 100 is intended for mounting on a DIN rail inside a closed cabinet. Due to its weight it should be installed on a strong DIN rail. No tool is required to install or remove the MFN 100.

Follow these steps to install the MFN 100 on a DIN rail:

1. Engage the DIN rail mounting clip of the MFN 100 with the upper edge of the DIN rail.
2. Push the MFN 100 down into the DIN rail.
3. Place the MFN 100 in a vertical position so that the mounting clip engages the lower edge of the DIN rail.

If you want to remove the MFN 100 from a DIN rail, follow these steps:

1. Push the MFN 100 down.
2. Rotate the MFN 100 upwards so that the lower edge of the DIN rail disengages.
3. Lift the MFN 100 slightly to remove it.

Setting up the MFN 100

You will need two network cables and a +24 V DC power supply for the setup. After mounting the MFN 100 on a DIN rail:

1. Connect pin 1 of the mating connector to +24 V DC.
2. Connect pin 2 of the mating connector to GND.
3. Plug the mating connector into the bottom side of the MFN 100.

4. Connect port 2 of the MFN 100 to a DHCP-enabled network with access to the Management System or internet access if the Management System is hosted by TTTech Industrial.

> **NOTE**
>
> If you are not sure how to allow external devices to connect to your network, please contact your IT administrator.

5. Plug in the power supply.

The MFN 100 will start after a few minutes and light up blue when all necessary services are initiated.

> **NOTE**
>
> - If you want to connect the MFN 100 to a fieldbus, connect a network cable to the I/O port of the MFN 100 and to your fieldbus interface.
> - You can also connect a second power supply to the MFN 100 as a backup. To do so, connect pin 3 of the mating connector to GND and connect pin 4 of the mating connector to +24 V DC.

Local UI

The local UI is provided by a web server that is running on the Nerve Device locally. Compared to the Management System, the local UI covers features that only concern the node itself. This chapter will cover:

- Network configuration
- Control of CODESYS applications

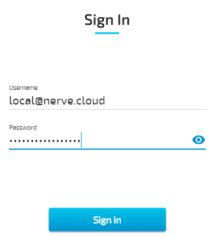
> **NOTE**
>
> The credentials for the local UI are set in the node registration process. If the product has been configured by TTTech Industrial, you can find the credentials in the customer profile that has been sent as part of the delivery.
>
> If you have not received a customer profile, please contact your sales representative or TTTech Industrial customer support at support@tttech-industrial.com.

Connecting to the Local UI

In order to access the local UI, you need to connect a workstation to the console port (P1) of the MFN 100 and configure the network adapter of your workstation. The IP address has to be in the

range from 172.20.2.5 to 172.20.2.254 with a 255.255.255.0 subnet mask. You also need the credentials for the local UI that you can find in the customer profile.

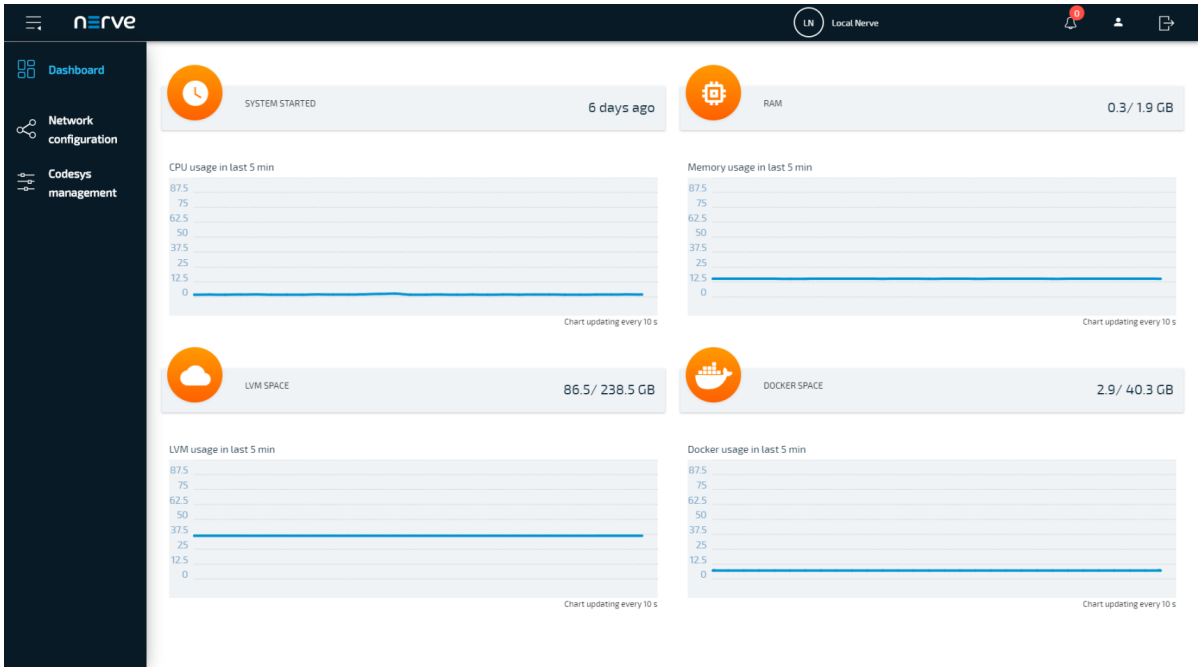
1. Follow this link to connect to the local UI: http://172.20.2.1:3000/
2. Log in with the credentials from the customer profile.



NOTE

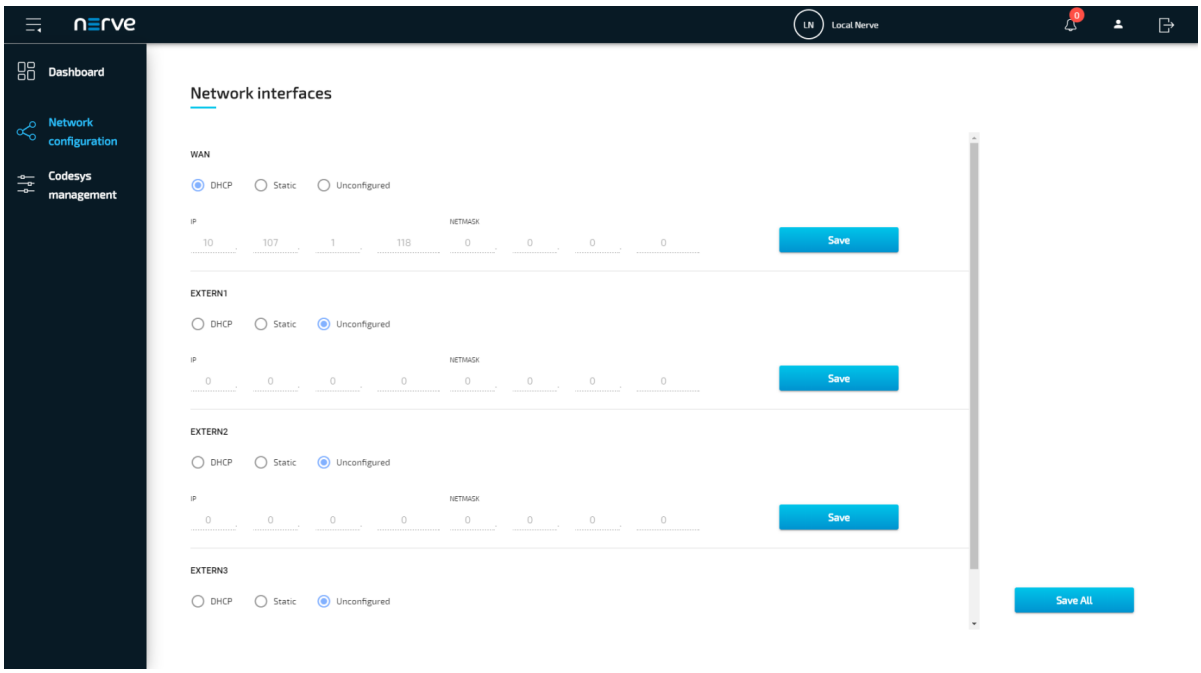
The link to the local UI above is part of the standard configuration. It is possible that the URL for your Nerve Device is different. You can find the URL that is valid for your Nerve Device in the customer profile.

You will reach the main page of the local UI.



Local Network Configuration

From the local UI, you can configure the Ethernet ports of the Nerve Device. The ports in the local UI represent ports 2, 3, 4 and 5 on the MFN 100. The console port (P1) and the I/O port of the MFN 100 are reserved and cannot be modified. The console port is used solely for configuration purposes. The I/O port is connected to the CODESYS runtime and used for fieldbus communication. Select **Network configuration** in the navigation on the left to reach this menu.



The options below are available for every interface:

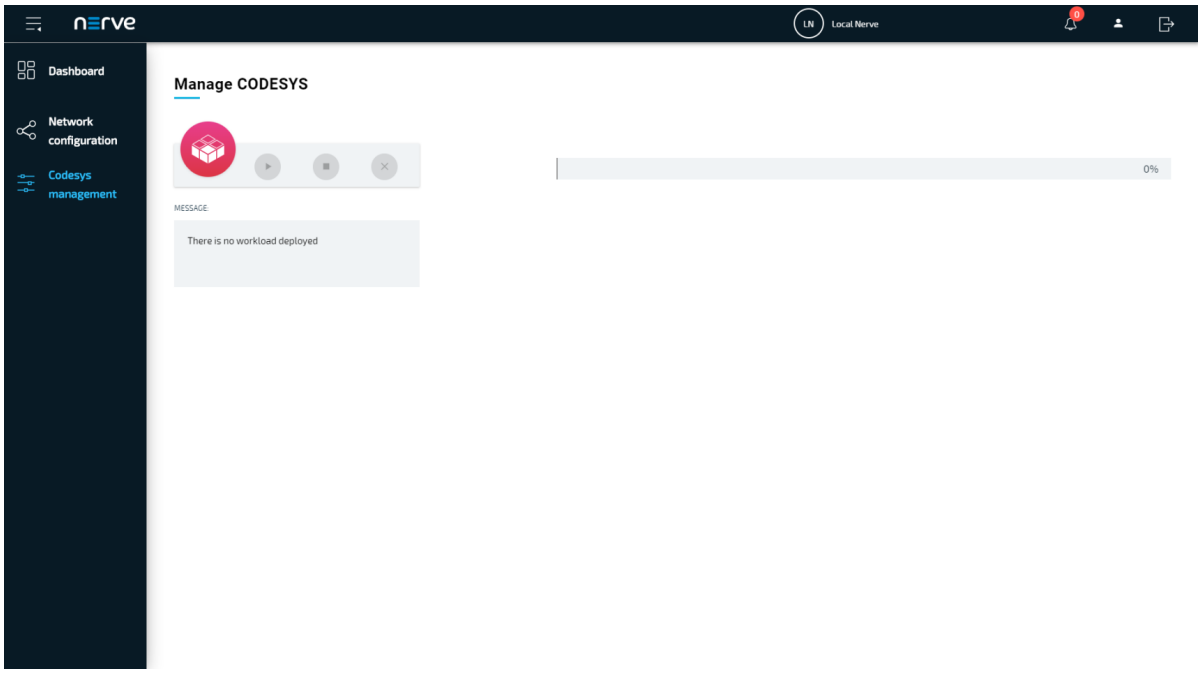
Item	Description
DHCP	The IP address of the port will be assigned by the DHCP server. If an IP address has been assigned, it will be displayed here.
Static	By selecting **Static**, you have to define the IP address of the port manually. Enter the IP address here to set a static IP address.
Unconfigured	If **Unconfigured** is checked, the port is disabled for the host but can still be used for virtual machines with bridged interfaces.

Network Interfaces

Interface Name	Physical Port on the MFN 100
WAN	Ethernet port 2 (P2)
EXTERN1	Ethernet port 3 (P3)
EXTERN2	Ethernet port 4 (P4)
EXTERN3	Ethernet port 5 (P5)

Control of CODESYS Applications

CODESYS workloads can only be controlled in the local UI, as operation of a CODESYS workload may have an impact on your machine operation and therefore should not be controlled remotely. Select **CODESYS management** in the menu on the left-hand side to reach the interface for controlling a CODESYS application running on the Nerve Device:



Function Name	Description
Start	This starts the CODESYS application.
Stop	This stops the CODESYS application and it is reset to its initial values.

Function Name	Description
Remove	This removes the CODESYS application from the Nerve Device. If you want to deploy the CODESYS application again, you have to do so directly with local access through the CODESYS Development System or through the Management System in the cloud.

NOTE

It takes a moment before CODESYS applications are actually started, stopped or removed.

Nerve Management System

The Nerve Management System is a web-based service that permits management of Nerve Blue nodes that are registered. It can be used to:

- Monitor nodes
- Deploy and control workloads on a node
- Manage workloads

NOTE

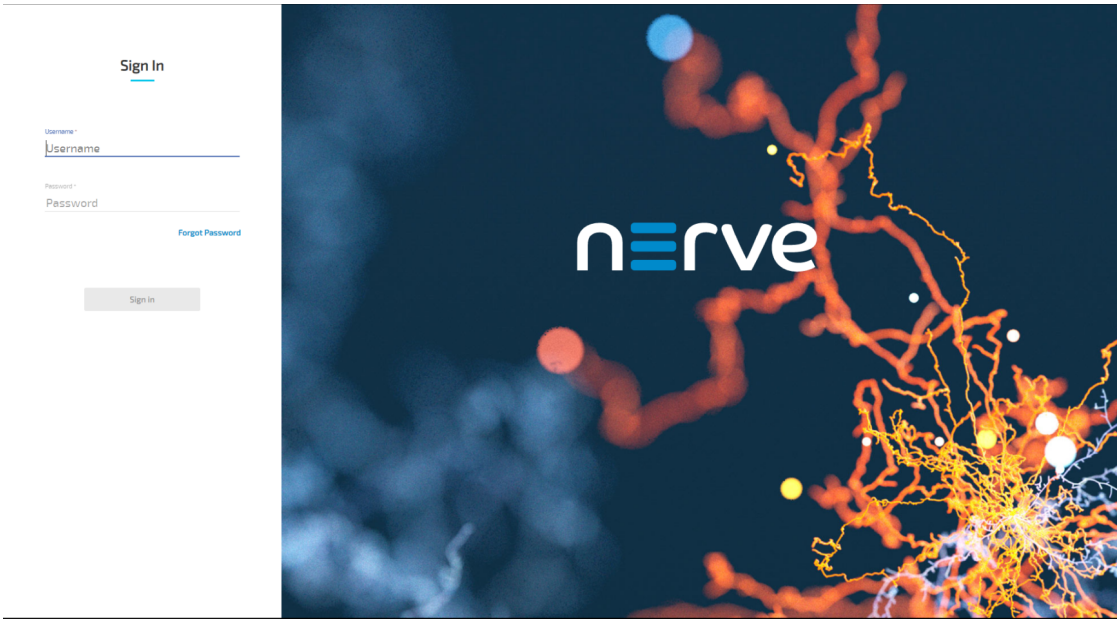
Google Chrome or Firefox Version 63 or later are recommended for the usage of the Management System.

Connecting to the Management System

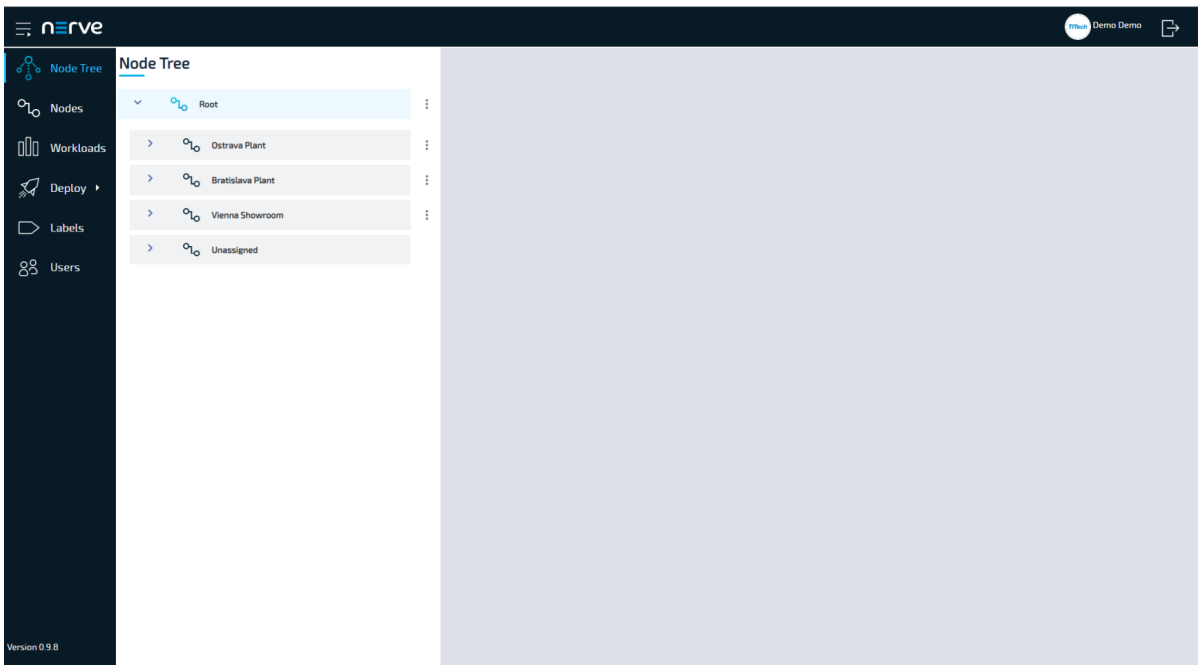
Please make sure that your Nerve Device is connected to the network through port 2 of the Nerve Device and that an IP address has been assigned by the DHCP server. If you need help with assigning an IP address, please contact your IT administrator.

The login credentials for the Management System are in the customer profile. The customer profile has been sent in form of a PDF as part of the delivery. If you have not received a customer profile, please contact your sales representative or TTTech Industrial customer support at support@tttech-industrial.com.

1. Go to the URL of the Management System in the customer profile.
2. Log in with the credentials provided in the customer profile.



You will reach the Node Tree in the Management System.



NOTE

There are no pre-configured tree elements on first login. All nodes will be located under **Root > Unassigned** by default.

From here you can manage nodes, provision workloads and deploy workloads among other options. The quick start guide mostly focuses on the provisioning and deployment of workloads. The user guide covers all options and settings available in the Management System.

NOTE

Port 443 (HTTPS) and port 8883 (MQTTS) of the corporate firewall have to be open for communication between nodes and the Management System.

First Steps with CODESYS and the MFN 100

This chapter will give you an introduction on how to start working with the integrated soft PLC in the MFN 100. First, some configuration and installation of files and libraries are required.

NOTE

- Please download the CODESYS Development System V3 from store.codesys.com for this chapter.
  We recommend version 3.5 SP14 (32 bit) or newer.
- Please connect your workstation to the console port (P1) of the MFN 100.

Installing the Device Descriptions

After you have downloaded and installed the CODESYS Development System on your workstation, you have to install the device description of the MFN 100 in the CODESYS Development System. The device description has the following filename:

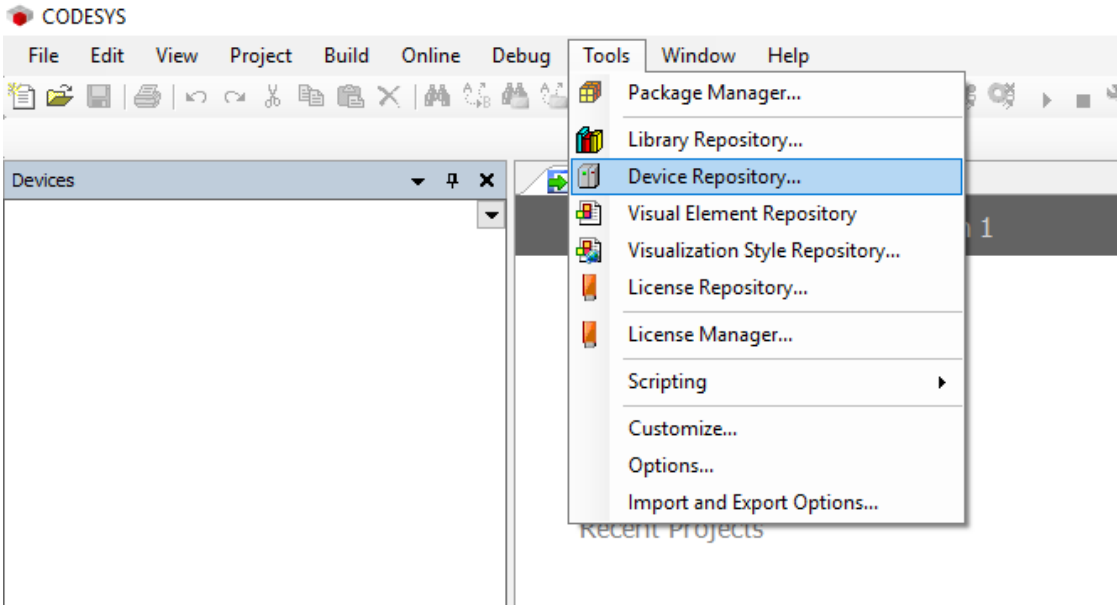
- Nerve_MFN_100_V3.5.XX.X.devdesc.xml

NOTE

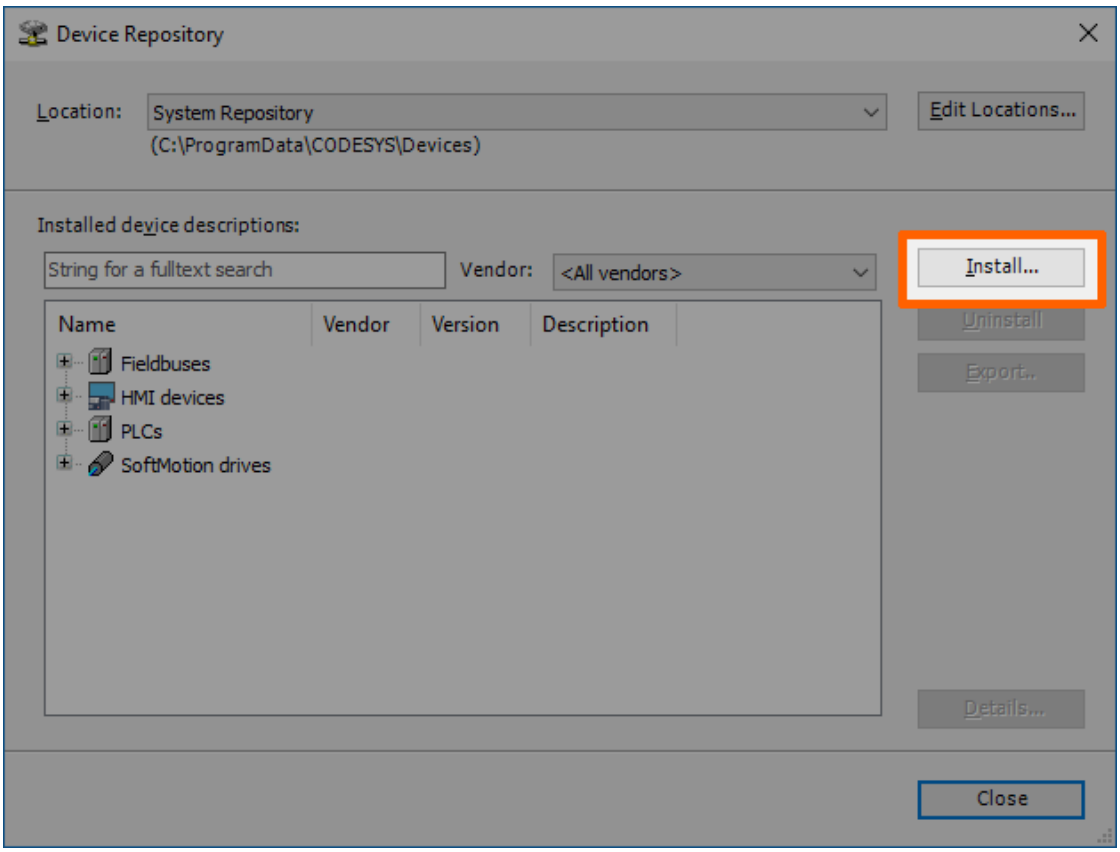
XX.X stands for the current version of the CODESYS Development System

The device description of the MFN 100 has been sent as part of the delivery. Please remember where you have saved the device description for the following steps.

1. Start the CODESYS Development System.
2. Go to **Tools > Device Repository**.



3. Click **Install**.



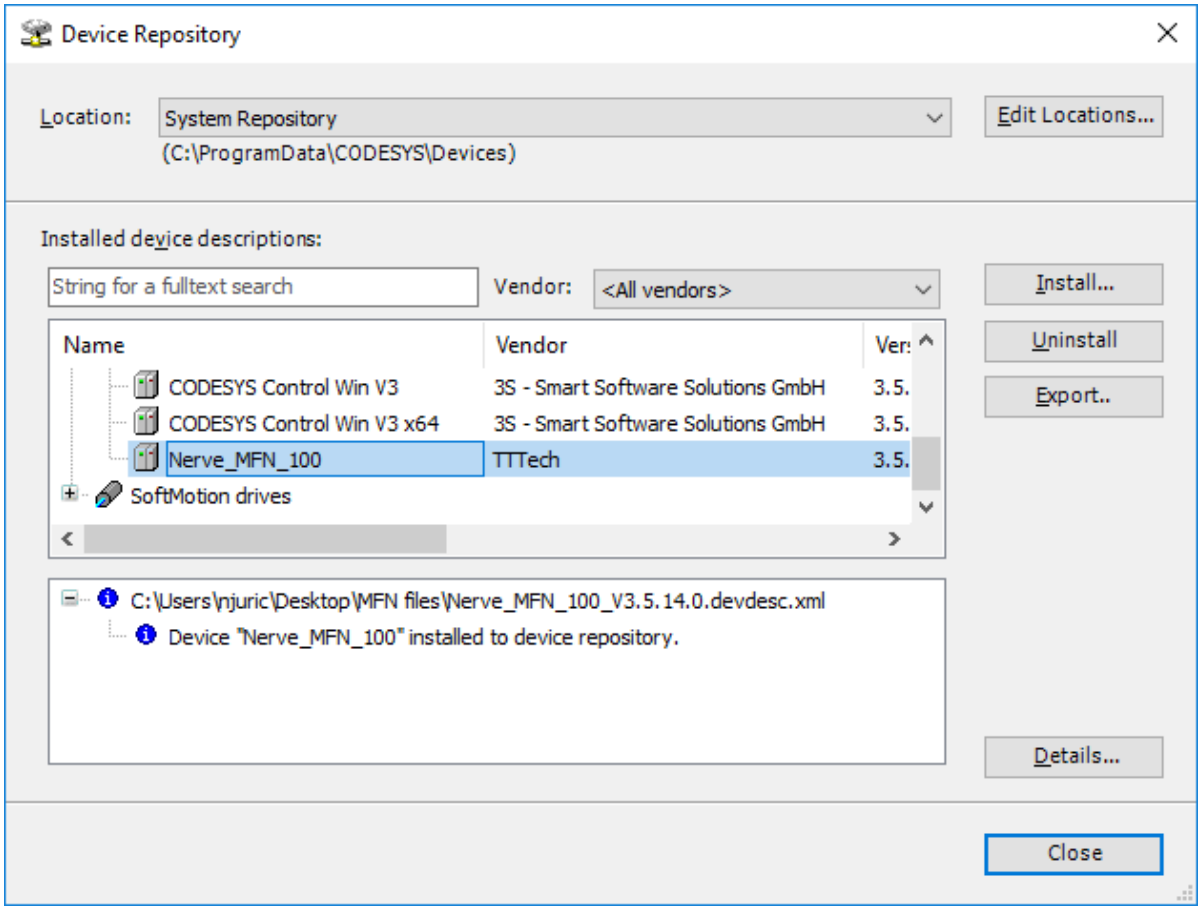
4. Go to the directory of the previously downloaded device description.

5. Select the device description of the MFN 100.

   The device description will look like this: Nerve_MFN_100_V3.5.XX.X.devdesc.xml

6. Click **Open**.

When the installation was successful, the MFN 100 will appear in the list of device descriptions in the middle of the window and you can close the window.



After installing the device description you can start working with the CODESYS Development System. However, libraries and device descriptions of generic devices might be missing so that the CODESYS Development System can work properly. The following chapters will walk you through the download process.

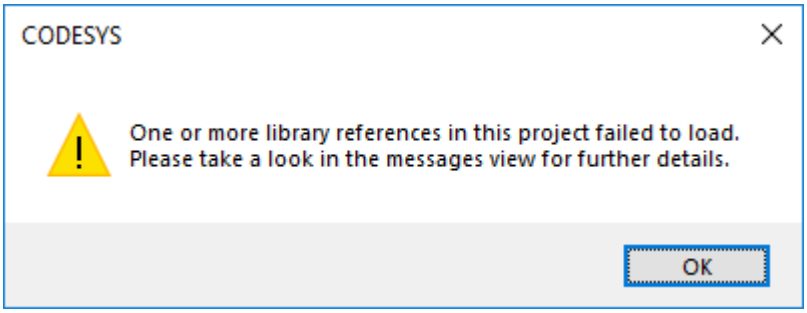
NOTE

If you have worked with the MFN 100 and the CODESYS Development System before, you might need to update the device description:

1. Follow the steps above to install the newest device description.
2. Right-click **Device (Nerve_MFN_100)** on the left side.
3. Select **Update Device...**.
4. Select the current device description in the new window.
5. Click **Update Device** in the lower-right.

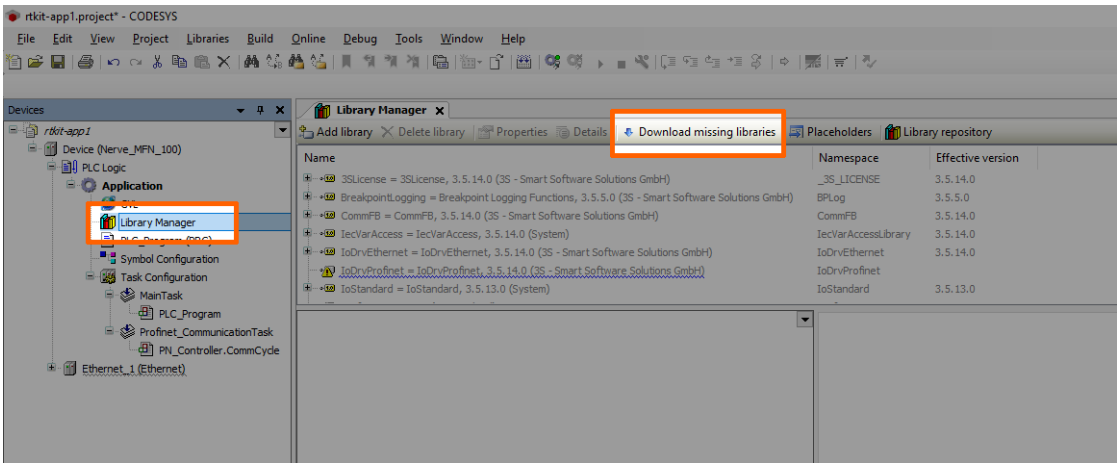
Downloading Missing Libraries

The error message for missing libraries might appear when you open or create a CODESYS project. The CODESYS Development System identifies the missing libraries automatically but you may have to repeat the following process a few times.

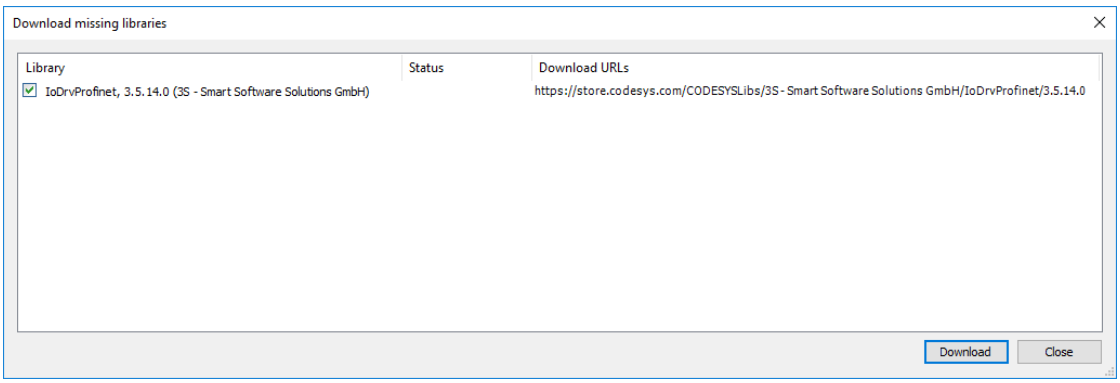


1. Open or create a CODESYS project.
2. If the error message about missing libraries appears, click **OK**.
3. Double-click **Library Manager** in the tree view on the left.

4. Click **Download missing libraries**.



5. Click **Download** in the new window.



6. Click **Close** when the download is finished.

7. Repeat steps 3 to 5 until no more libraries appear in the download window.

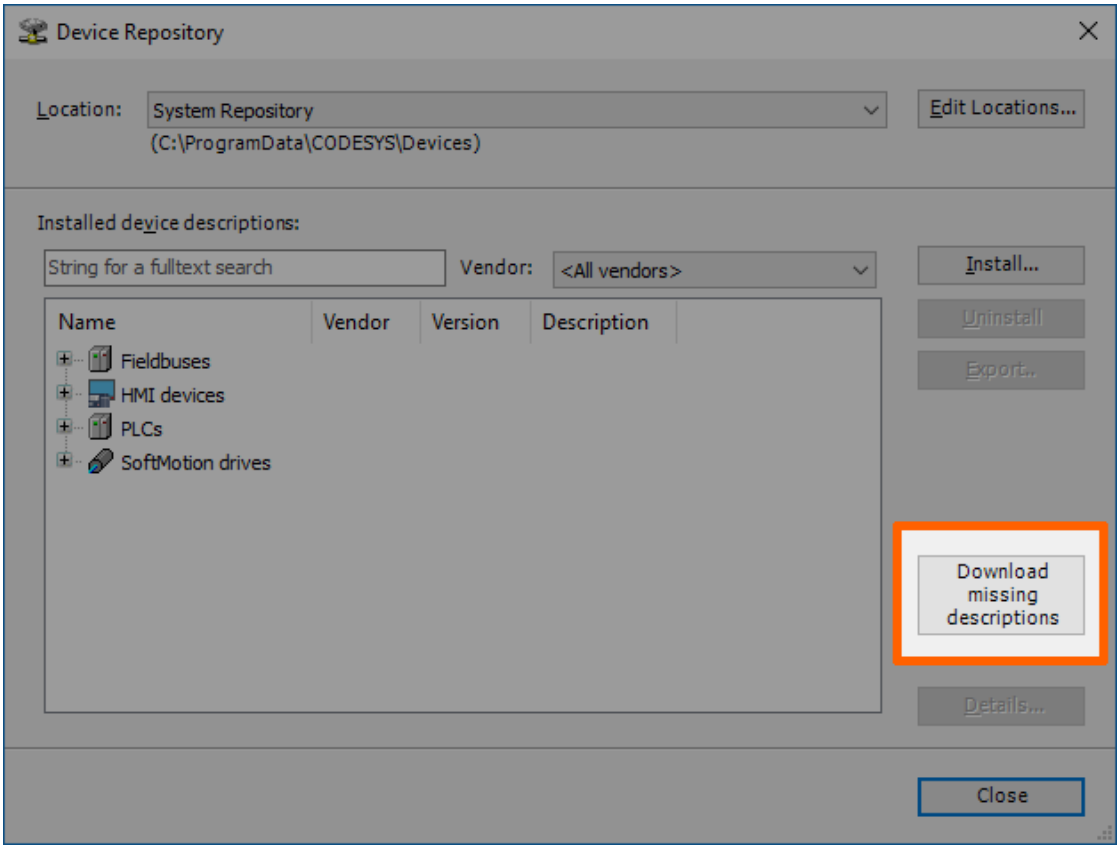
Downloading Missing Device Descriptions

Apart from the device description for the MFN 100 that you have installed manually before, device descriptions of generic devices may be missing for the CODESYS Development System to function as intended. The CODESYS Development System will identify the missing device descriptions

automatically but this time it will not generate an error message unless you try to load a CODESYS application into the MFN 100.

1. Click **Tools > Device Repository**.

2. Click **Download missing descriptions**.



NOTE

If no device descriptions of generic devices are missing, the button for downloading missing descriptions will not appear.

3. Click **Download** in the new window.

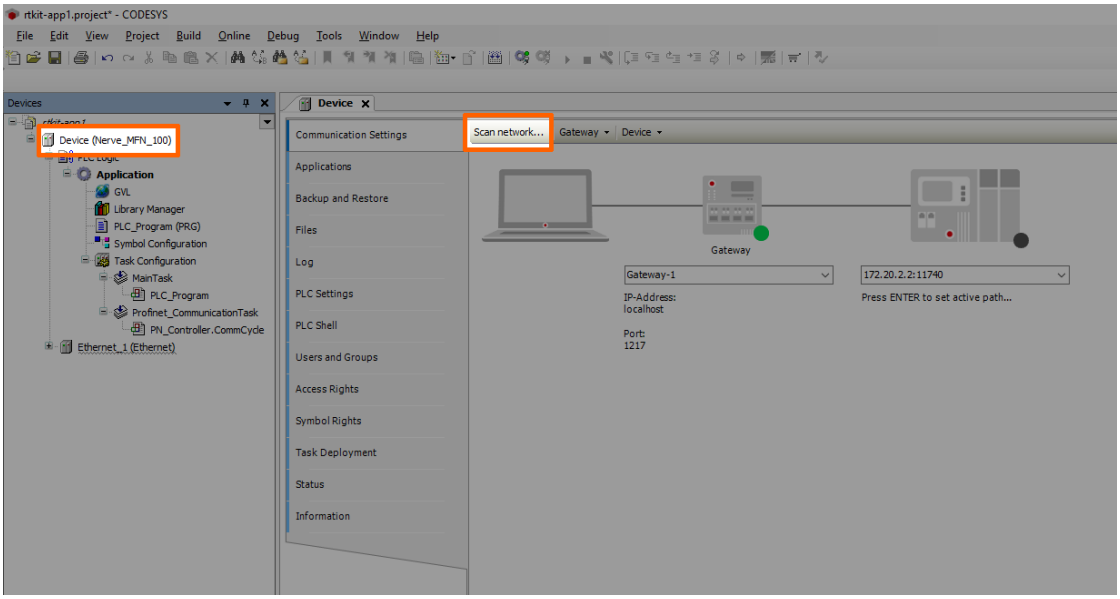
4. Click **Close** when the download is finished.

Connecting to the MFN 100

Before you can download CODESYS applications to the MFN 100, please make sure that the device description of the MFN 100 is installed in the CODESYS Development System.

1. Open or create a CODESYS project.
2. Double-click **Device (Nerve_MFN_100)** in the tree view on the left.

3. Go to **Communication Settings > Scan network...**.



4. Select the MFN 100 (here **nerve-rtvm [XXXX.XXXX]**) in this window.

   !nerve-rtvm

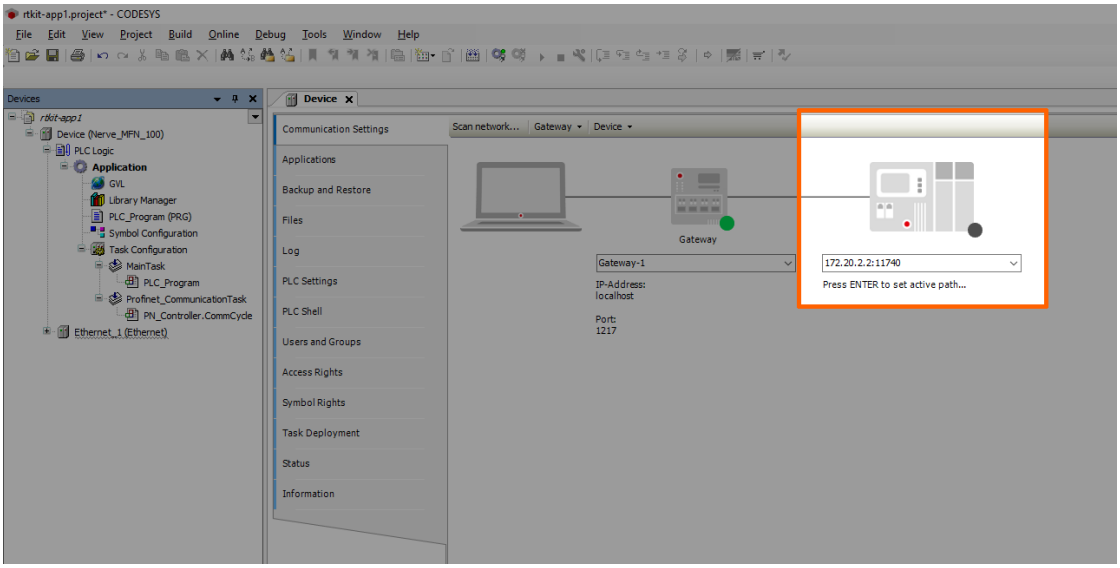
> **NOTE**
>
> When more than one network is active on your workstation, it sometimes happens that the MFN 100 cannot be found. Please continue reading if the MFN 100 does not appear in this window.

5. Click **OK**.

Typically the MFN 100 will be found automatically. If the MFN 100 cannot be found, you have to enter the IP address and port of the CODESYS runtime manually.

1. Double-click **Device (Nerve_MFN_100)** in the tree view on the left.
2. Go to **Communication Settings** in the middle of the window.

3. Enter 172.20.2.2:11740 in the text box under the device on the right.



4. Press Enter.

The CODESYS Development System is now connected to the MFN 100 and you can download applications into the CODESYS runtime.

Downloading an Application to the MFN 100

You can load CODESYS applications directly into the MFN 100. However, before you can download an application into the MFN 100 it needs to be free of errors.

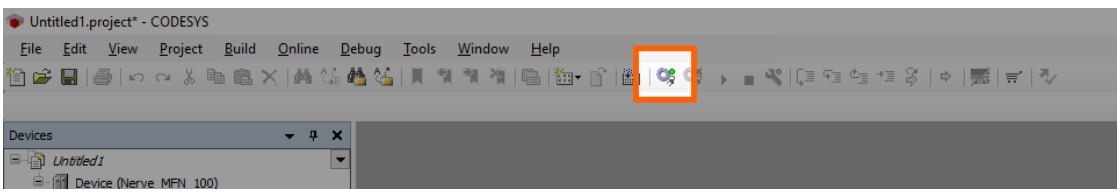
The process of downloading an application is slightly different if you are downloading an entirely new application into the MFN 100 or if you are updating an application that has already been downloaded into the MFN 100. If you are updating an application that you have downloaded to the MFN 100 before, please continue with Downloading an Updated Application to the MFN 100 further down below.

Downloading a New Application to the MFN 100

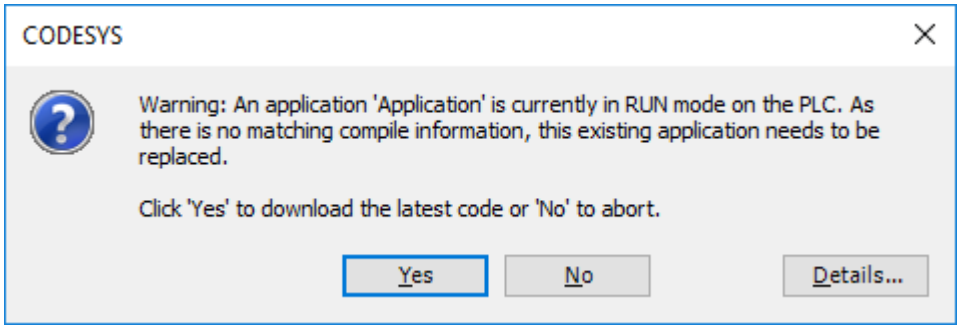
Once you have created a project and you are finished with programming, you can download the CODESYS application to the MFN 100 directly.

1. Open the CODESYS project you want to load into the MFN 100.

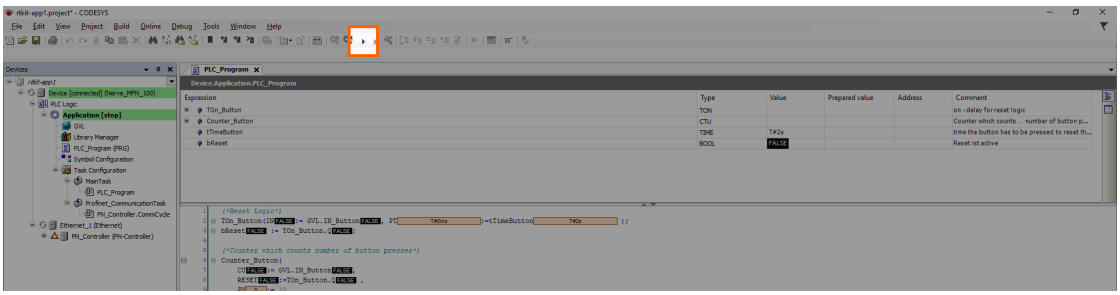
2. Click the **Login** symbol in the CODESYS menu bar.



3. Click **Yes** in the pop-up window.



4. The application is stopped now. Click the **Play** symbol in the CODESYS menu bar.



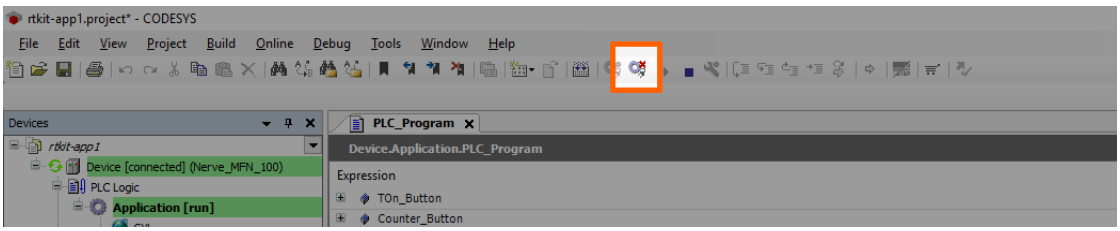
The application is now loaded to the MFN 100.

Downloading an Updated Application to the MFN 100

If you update an application after you have loaded it into the MFN 100, you need to download it into the MFN 100 again. The download process is slightly different from downloading a new application into the MFN 100.

1. Stop the CODESYS application that you have loaded into the MFN 100 through the local UI.

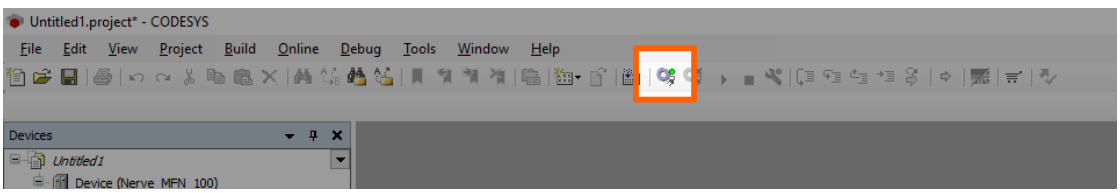
2. Click the **Logout** button in the CODESYS toolbar.



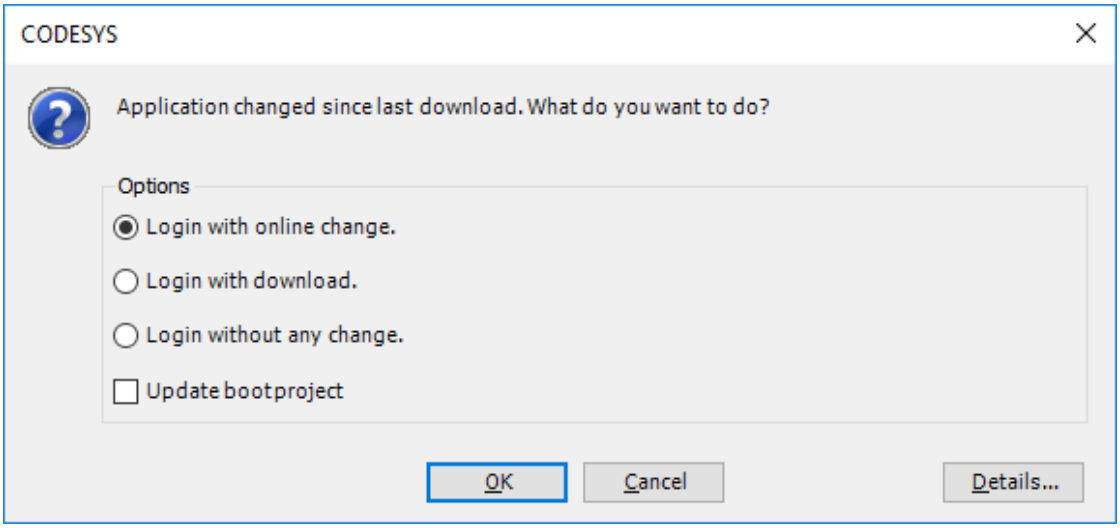
3. Expand **Device (Nerve_MFN_100) > PLC Logic > Application**.

4. Double-click **PLC_Program (PRG)**.
5. Perform your changes.

6. Click the **Login symbol** in the CODESYS menu bar.



7. In the pop-up window, select one of the options.



Item	Description
Login with online change.	The updated application will be loaded into the MFN 100. Variable values will not be reset. If the application was running before, it will be running after the download.
Login with download.	The updated application will be loaded into the MFN 100. Variable values will be reset. The application is stopped.
Login without any change.	The updated application will not be loaded into the MFN 100 but the code will keep your changes.

8. Click **OK**.

The application is now loaded to the MFN 100.

NOTE

For more help with programming PLC applications in the CODESYS Development System go to help.codesys.com.

Registering a Node in the Management System

In this version, it is required to register the node through the command line before you can use it in the Management System. The command line tool will be executed locally on the Nerve Device. The files have been sent as part of the delivery.

NOTE

If you have not received the required files for the registration of a node, please contact your sales representative or TTTech Industrial customer support at support@tttech-industrial.com.

Hardware Setup

In order to connect the node to the Management System, you will need to set up the Nerve Device and connect your local workstation to the Nerve Device:

1. Please set up your Nerve Device as described in Hardware Setup.
2. Connect your workstation to the console port (P1) of the MFN 100.

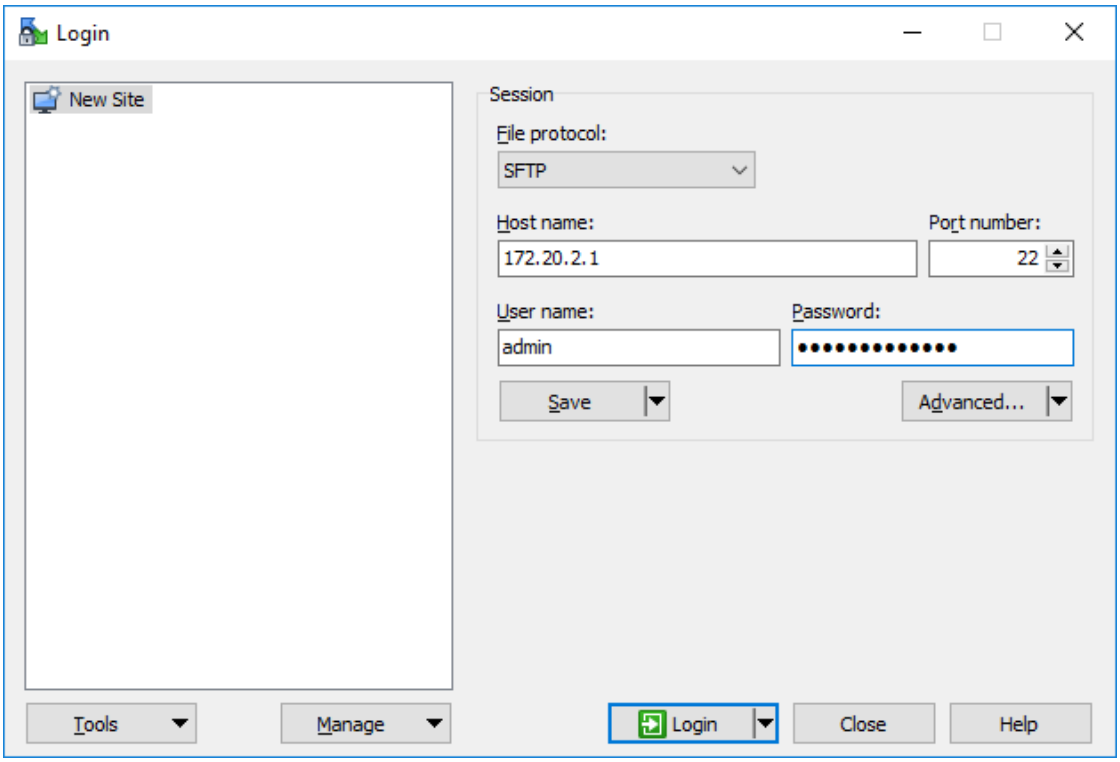
With this, the hardware is ready for the registration process.

> **NOTE**
>
> Port 443 (HTTPS) and port 8883 (MQTTS) of the corporate firewall have to be open for communication between nodes and the Management System.

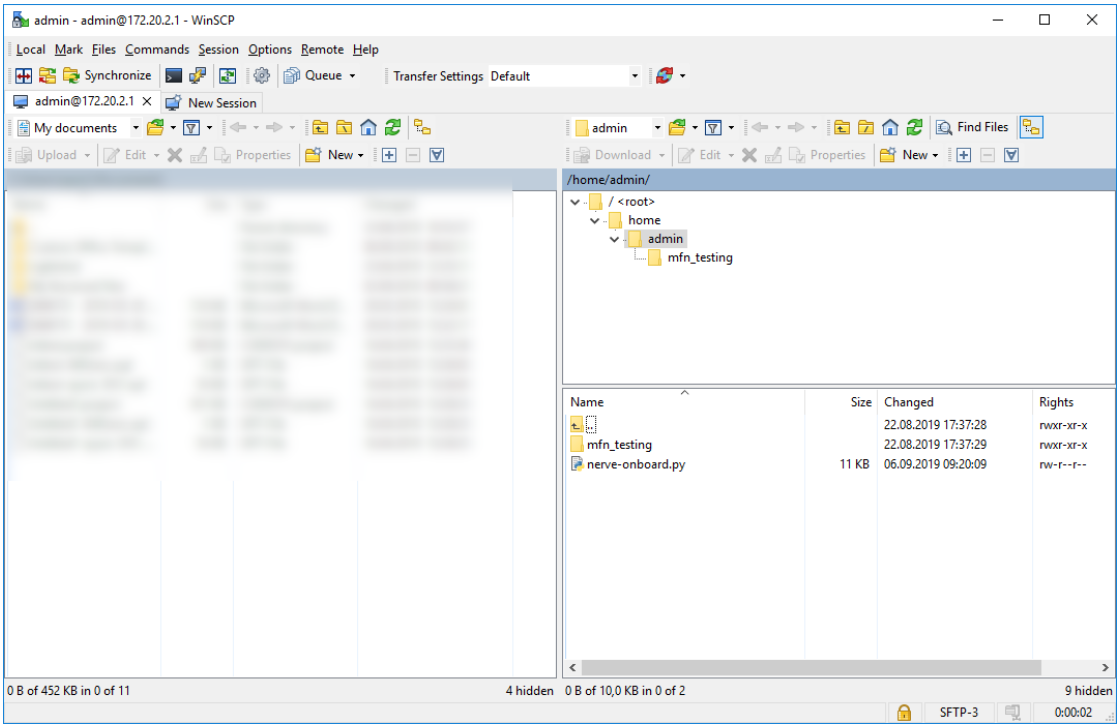
Copying the Tool to the Nerve Device

As the dependencies for the command line tool are already installed on the Nerve Devices, you only need to copy it to the Nerve Device and execute it from there. To do so, you will need an SSH client and a file transfer client. PuTTY and WinSCP are used as examples.

1. Open a file transfer client like WinSCP.
2. Enter 172.20.2.1 under **Host Name**.
3. Enter the credentials for host access to the Nerve Device from the customer profile below under **User name** and **Password**.



4. Copy the nerve-onboard.py to the Nerve Device. The default path is /home/admin/ on the Nerve Device.

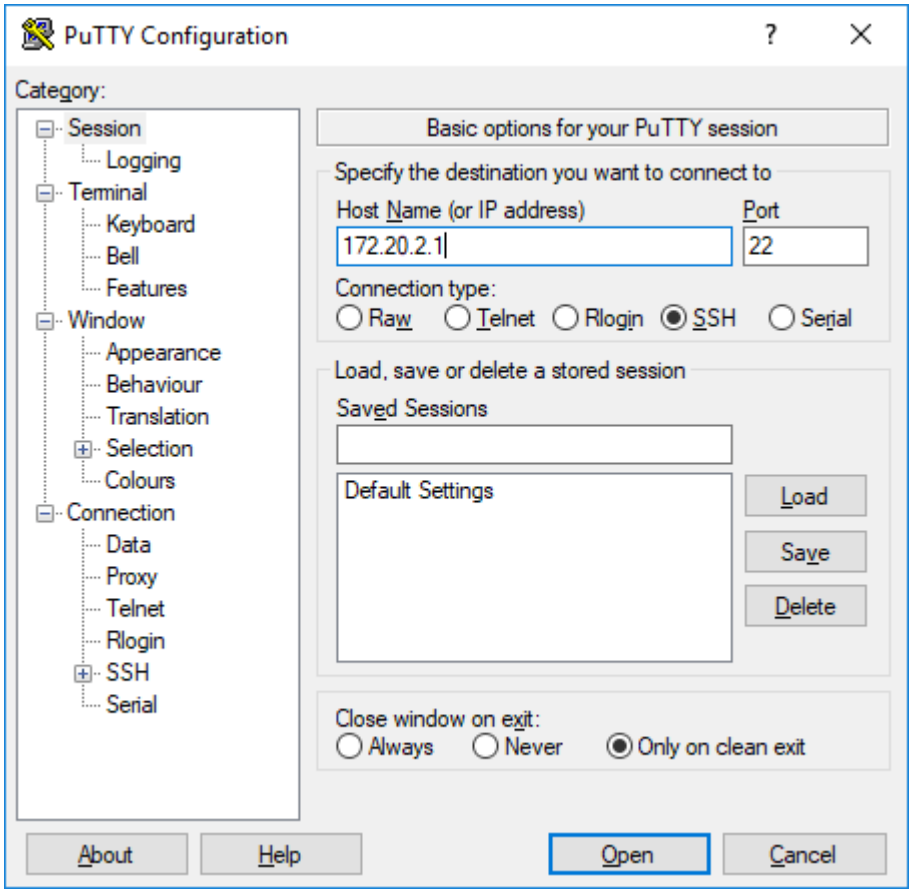


With the command line tool now copied to the Nerve Device, you can execute the tool on the Nerve Device. We will use an SSH connection to do so.

Using the Tool

Now that all preparations are complete, you can execute the command line tool and start the registration of the node in the Management System.

1. Open an SSH client like PuTTY.
2. Enter 172.20.2.1 under **Host Name (or IP address)** to log in to the host of the Nerve Device.



3. Click **Open**.

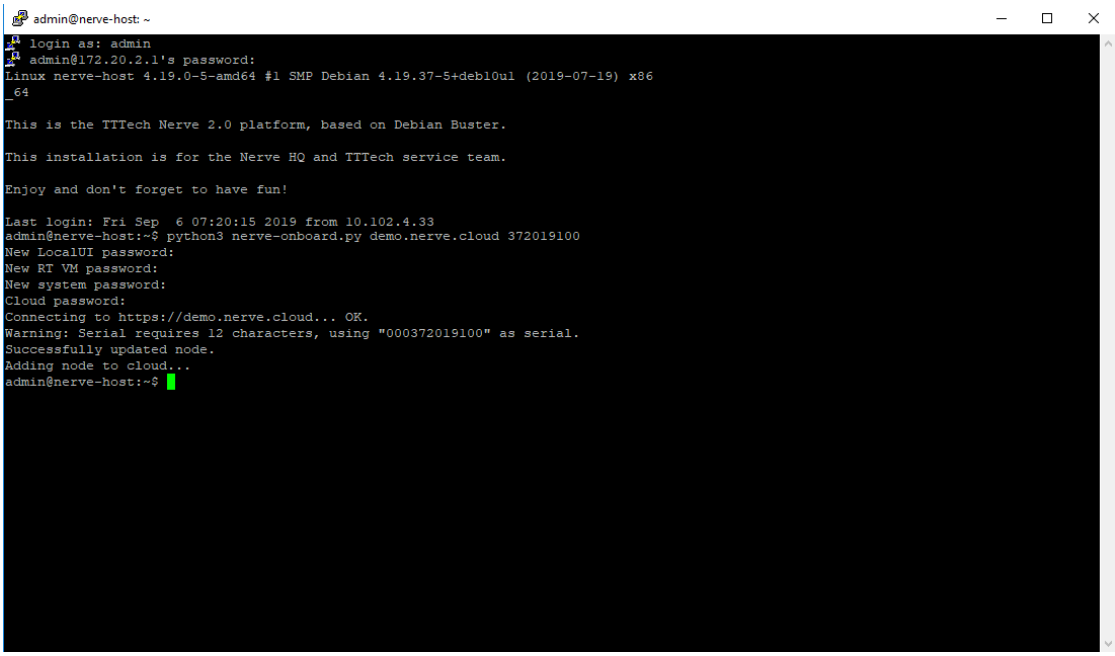
4. Log in with the credentials for host access of the Nerve Device from the customer profile.

5. Enter python3 nerve-onboard.py <domain> <serial> and replace the placeholders with this information:

Item	Description
<domain>	This is the URL of your cloud. Please enter the URL without the protocol in the beginning. You can find it in the customer profile. Example: example.nerve.cloud
<serial>	Enter the serial number of your Nerve Device here. You can find it on the label of the Nerve Device.

6. Enter the following information in the table. The tool will prompt you one by one:

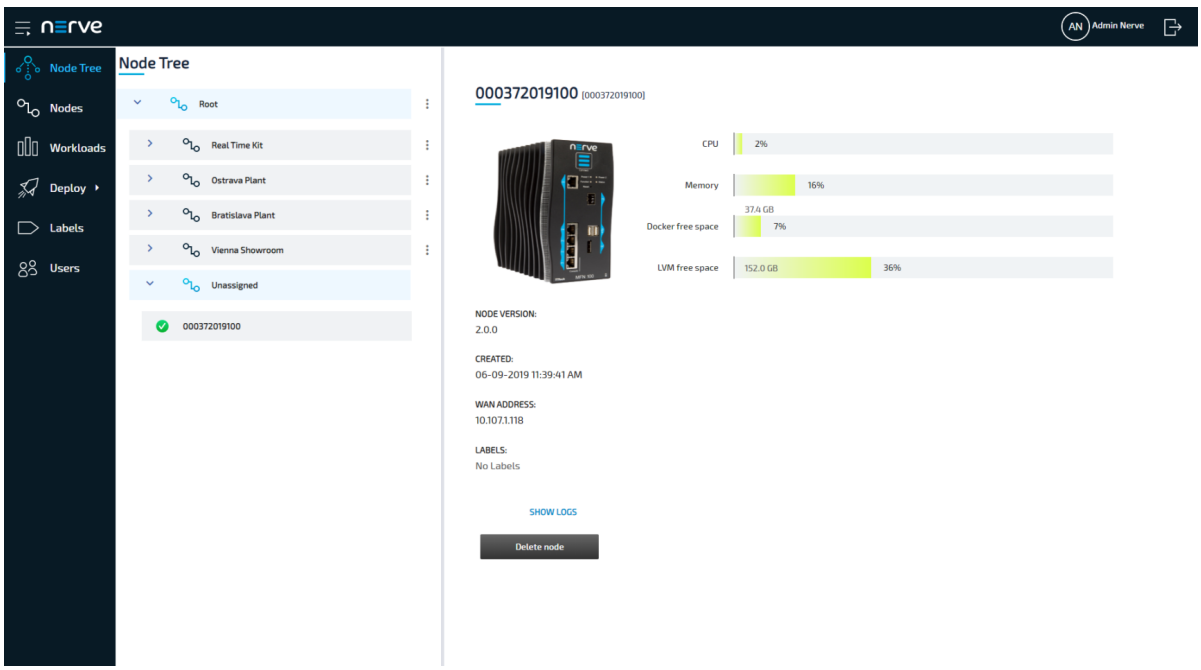
Item	Description
New LocalUI password	Define the new password for the local UI of your Nerve Device.
New RT VM password	Define the new password for the CODESYS runtime.
New system password	Define the new password for SSH host access.
Cloud password	Enter the password of your cloud access from the customer profile.



NOTE

The passwords above must be at least 8 characters long.

The node is now online and registered in the Management System. You can find it under **Root > Unassigned** in the Node Tree.



NOTE

Please make sure to note down the new passwords you have defined as you will need them later.

Working with CODESYS Applications

In order to work with CODESYS applications on nodes, workloads need to be provisioned in the Management System. Here, provisioning is the creation of a workload and its storage in the workload repository of the Management System so that it can be deployed to nodes. This requires configuration of the workload and files that need to be uploaded to the Management System. After that, the workload can be deployed to nodes.

Before the workload can be provisioned, however, a CODESYS application has to be loaded into the CODESYS runtime first. Refer to the introduction to working with CODESYS and the MFN 100 first before continuing.

Once a CODESYS application has been loaded into the MFN 100, the following steps have to be taken:

1. Creating the ZIP file of the CODESYS application
2. Transferring the ZIP file to a local workstation

In this version, Nerve Blue does not provide a GUI based method for creating a ZIP file of the CODESYS application. Therefore, this chapter focuses on the manual process. Two tools are required for the instructions below, assuming Windows is used on the workstation:

- an SSH client like PuTTY
- a file transfer client like WinSCP

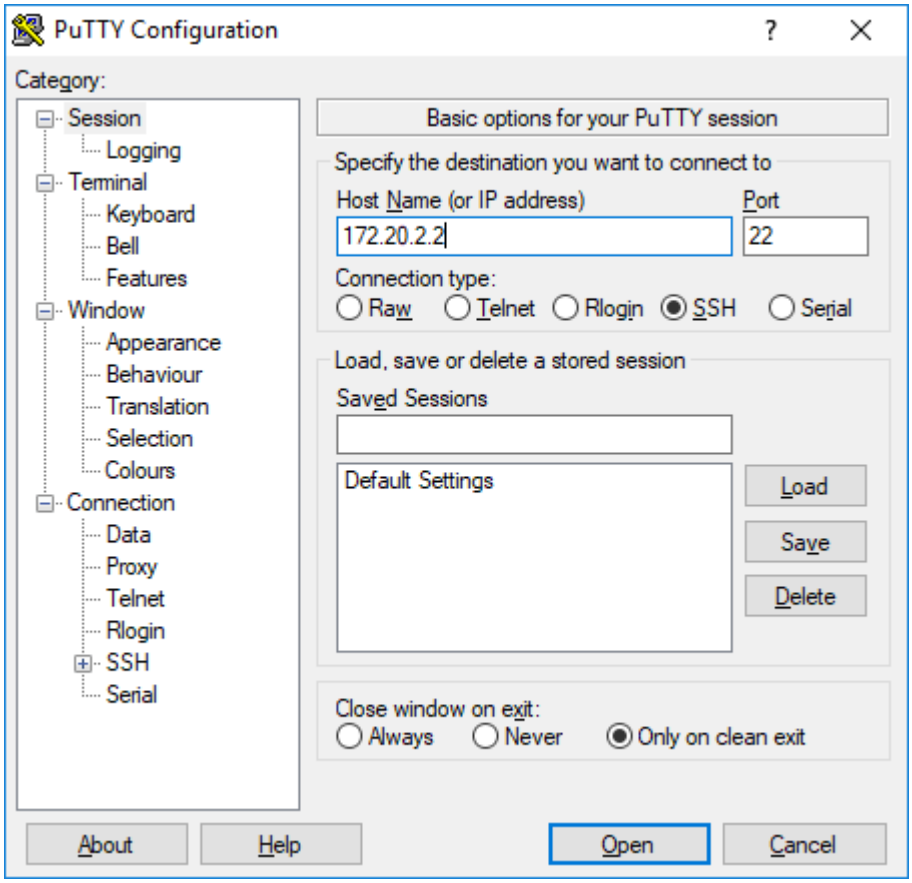
Also the workstation needs to be connected to the console port (P1) of the MFN 100 and the network adapter of the workstation needs to be configured. The IP address has to be in the range from 172.20.2.5 to 172.20.2.254 with a 255.255.255.0 subnet mask.

Creating the ZIP File on the Nerve Device

First, the CODESYS project needs to be zipped on the Nerve Device before it can be copied from the CODESYS runtime.

1. Open an SSH client like PuTTY.

2. Enter 172.20.2.2 under **Host Name (or IP address)** to log in to the CODESYS runtime of the MFN 100.



3. Click **Open**.
4. In the new window, enter the admin password for the CODESYS runtime.
5. Enter the following commands:

   sudo su

   systemctl stop nerve-codesys

   cd/var/lib/codesys

   zip -r /tmp/<userproject>.zip CmpApp.cfg PlcLogic/

> **NOTE**
>
> You can freely name the ZIP file here by replacing <userproject> with a name of your choice.

The ZIP file has now been created and is available on the Nerve Device.

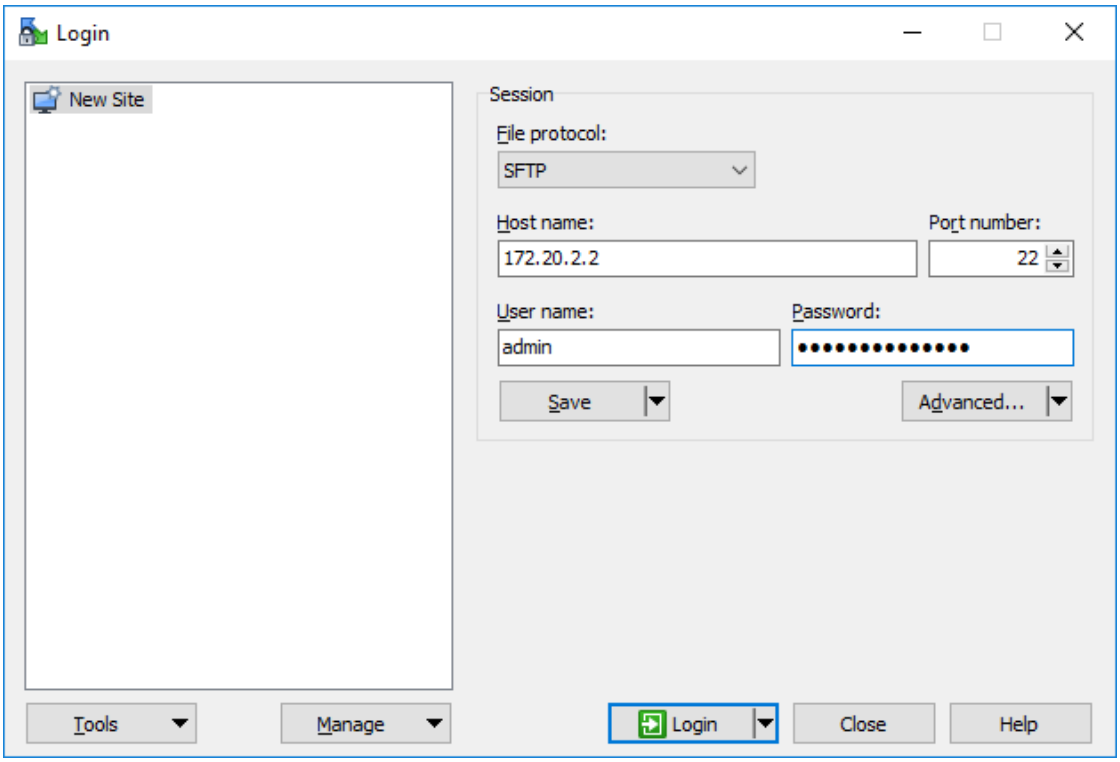
Transferring the ZIP File to a Local Workstation

After zipping the CODESYS project on the Nerve Device, you need to transfer it with a file transfer client to your workstation.

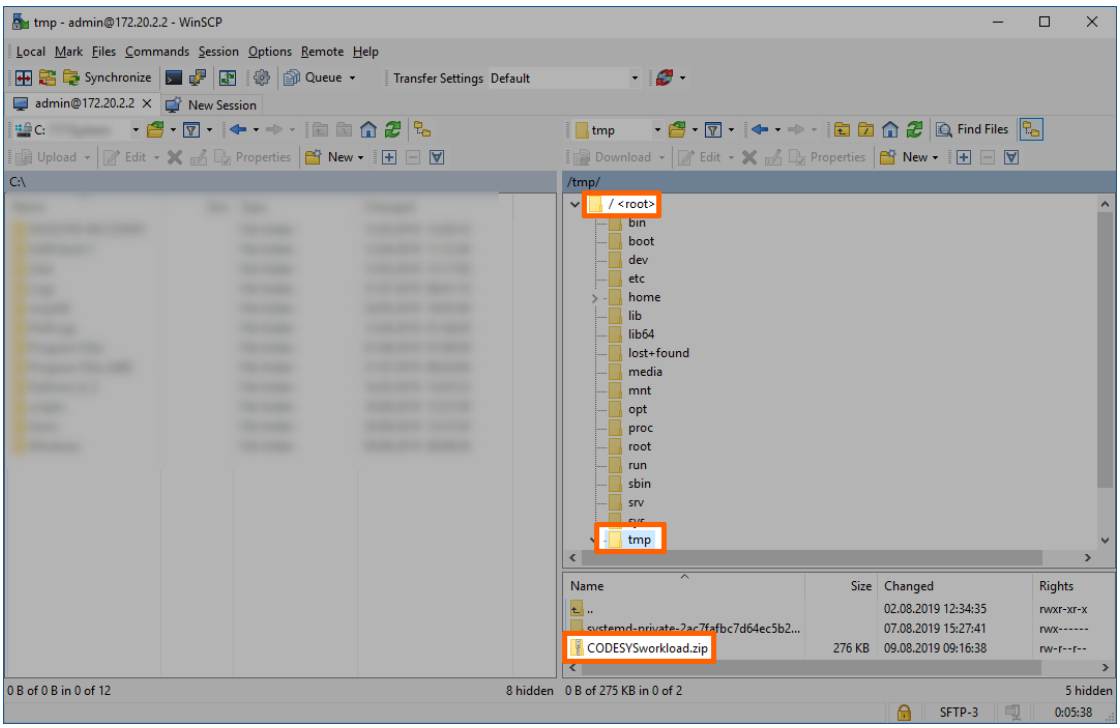
1. Open a file transfer client like WinSCP.

2. Enter 172.20.2.2 under **Host Name**.

3. Enter the credentials for the CODESYS runtime below under **User name** and **Password**.



4. Navigate to the **/tmp/** directory on the right side of the WinSCP window. You will find it in the **root** directory.



5. Drag and drop the ZIP file to your workstation.

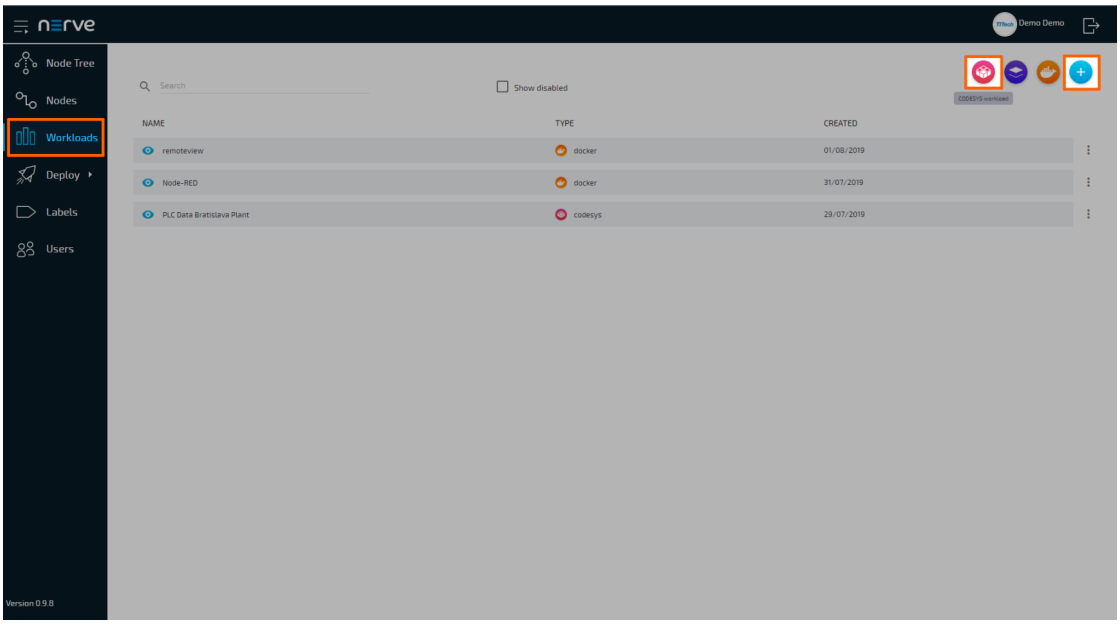
Since the CODESYS project ZIP file is on your local workstation, you can now provision a CODESYS workload in the Management System.

Provisioning a CODESYS Workload

The following instructions cover the basic requirements for provisioning a CODESYS workload. Optional settings will be left out. Extended options are addressed in the last section of this chapter.
There are two further types of workloads that can be provisioned: Virtual Machine workloads and Docker workloads. The process for each workload is highlighted in its respective chapter.

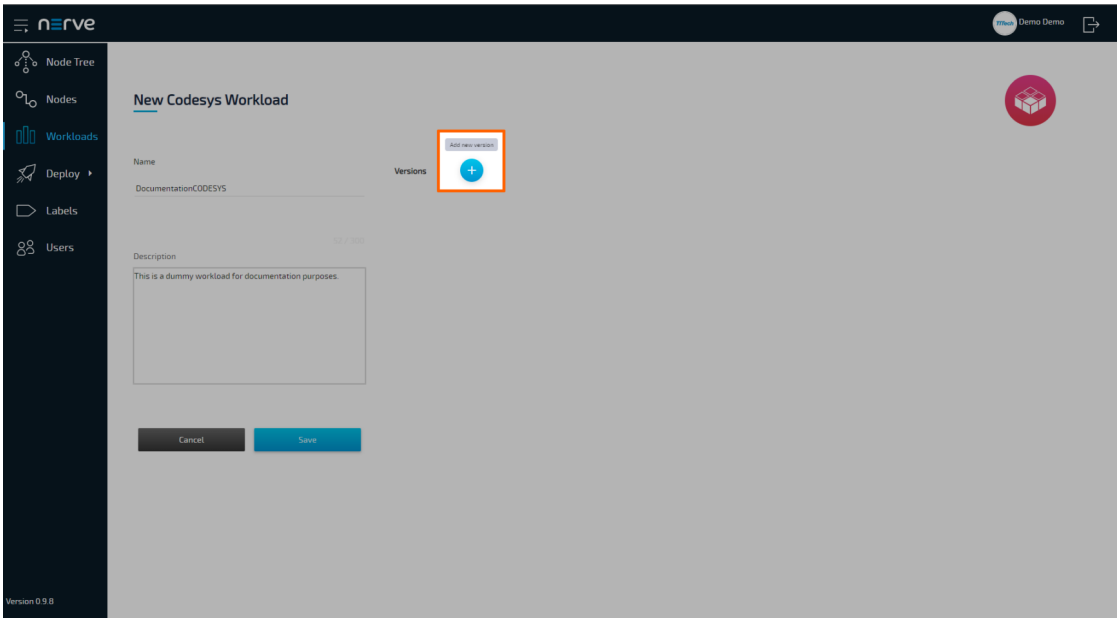
1. Log in to the Management System.
2. Select **Workloads** in the left-hand menu.
3. Select **+** in the upper-right corner.

4. Select the CODESYS symbol (**CODESYS workload**) on the left of the three symbols that expanded.



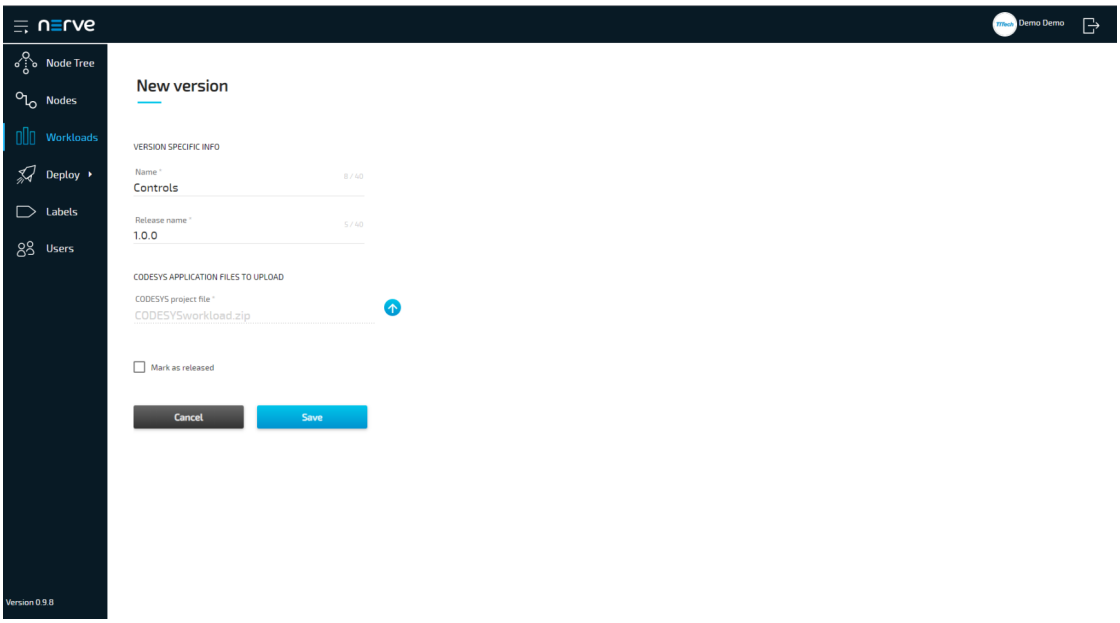
5. In the new window, enter a name for your workload.

6. Select **+** next to **Versions** to add a new version of the workload.



7. In the new window, enter the following information:

Item	Description
Name	Enter a **Name** for the version of this workload.
Release name	Enter a **Release name** for the version of this workload.
CODESYS project file	Click the **upward arrow** symbol to add the CODESYS application ZIP file. This is the ZIP file that you have created before.



8. Click **Save**.

The workload has now been provisioned and is ready to be deployed in the **Deploy** menu.

Settings for CODESYS Workloads

In the instructions above, optional settings have been left out. Below is an overview of all options with an explanation to each option.

Setting	Description
VERSION SPECIFIC INFO	**Name** A name for your workload version. Choose a precise name to make the workload version unambiguous. **Release name** A release name for your workload version. This could be a version number. Example: 1.0.1
CODESYS APPLICATION FILES TO UPLOAD	Here you have to upload the **CODESYS project file**. This is a ZIP file that has to be generated from a CODESYS project running in the CODESYS runtime. Upload it here by clicking the **upward arrow** symbol to open your file browser.
SELECTOR	**Labels** If you have defined labels and assigned them to nodes, you can add them as selectors to the workload. When deploying a workload, the list of nodes will be filtered automatically to the specified label.
Mark as released	Tick this checkbox if you want to mark this workload as released. Once marked as released, the workload cannot be edited anymore.

Working with Virtual Machines

In order to work with virtual machines on nodes, workloads need to be provisioned in the Management System. Here, provisioning is the creation of a workload and its storage in the workload repository of the Management System so that it can be deployed to nodes. This requires configuration of the workload and files that need to be uploaded to the Management System. After that, the workload can be deployed to nodes.

Before the workload can be provisioned, however, it is required to set up the virtual machine on the node. In more detail, what needs to be done is:

1. Creating a virtual machine on the node
2. Installing an operating system on the virtual machine
3. Obtaining the virtual machine IMG and XML files
4. Provisioning the Virtual Machine workload in the Management System

In this version, Nerve Blue does not provide a GUI based method for installing an OS on a virtual machine and obtaining the virtual machine IMG and XML configuration files. Therefore, this chapter focuses on the manual process. Three tools are required for the instructions below, assuming Windows is used on the workstation:

- an X Server application like Xming
- an SSH client like PuTTY
- a file transfer client like WinSCP

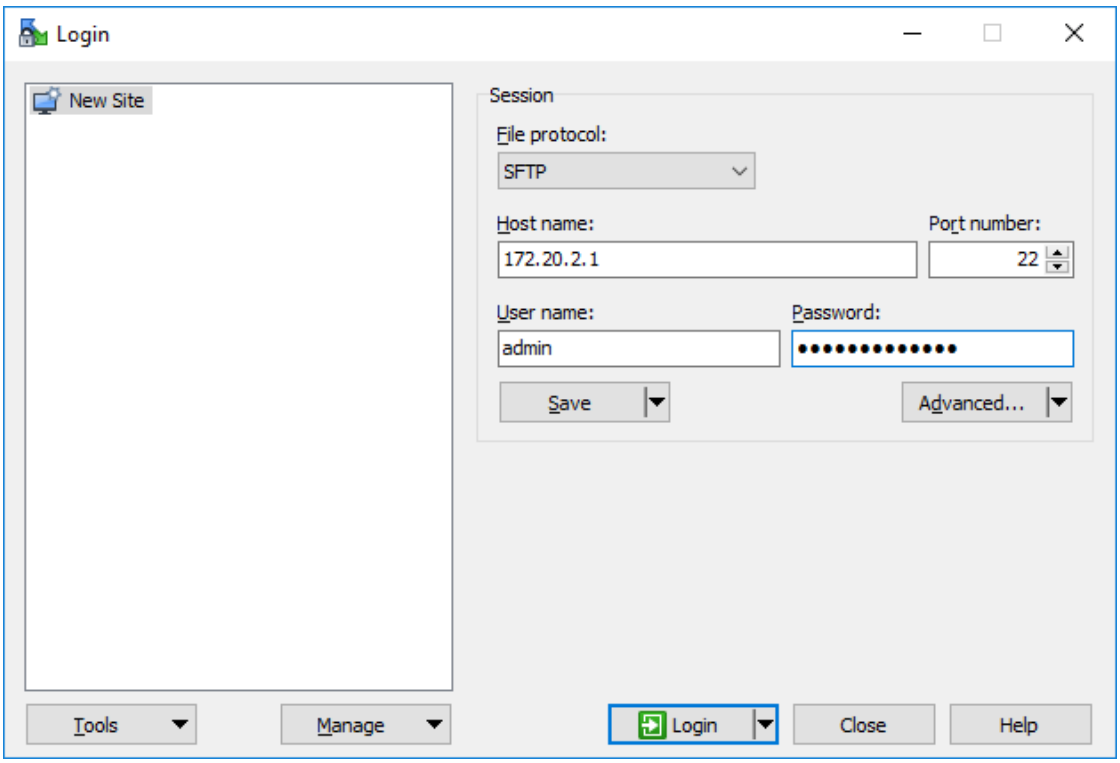
Also the workstation needs to be connected to the console port (P1) of the MFN 100 and the network adapter of the workstation needs to be configured. The IP address has to be in the range from 172.20.2.5 to 172.20.2.254 with a subnet mask of 255.255.255.0. The ISO file of the OS that will be installed on the virtual machine is also required.

The instructions below are split up into multiple parts to make them easier to follow. The subsections of the instructions are connected and every subheading is a requirement for the next paragraph.

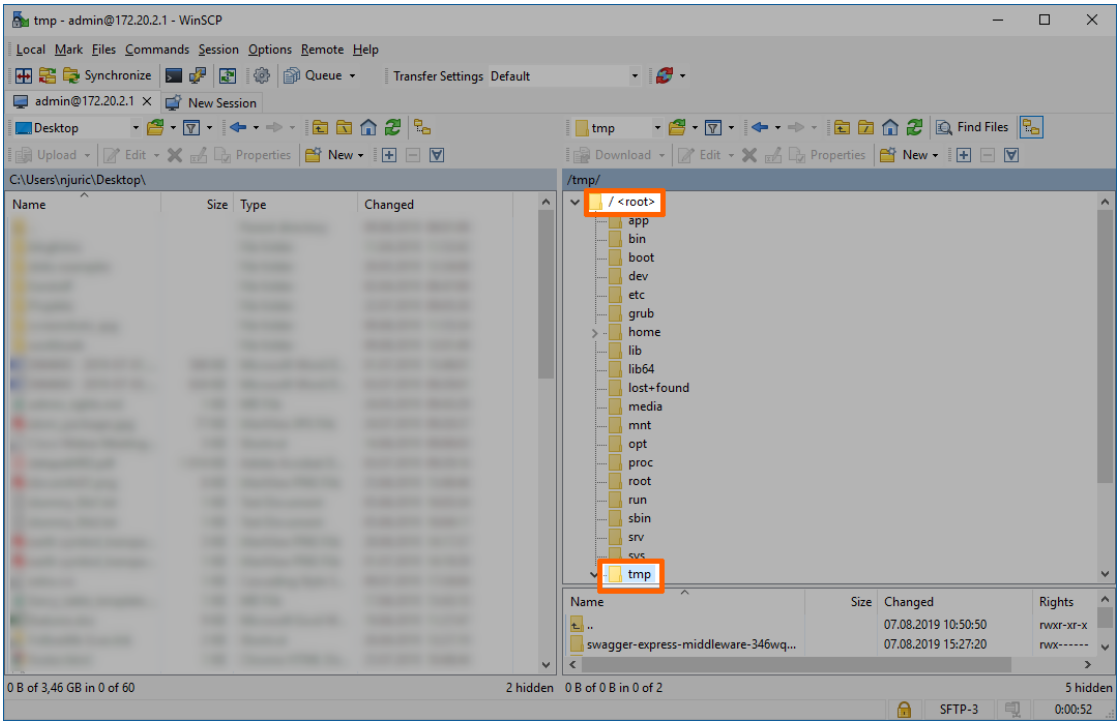
Copying the ISO File to the Nerve Device

First, it is necessary to copy the ISO file of the OS to the Nerve Device for the installation on the virtual machine.

1. Open a file transfer client like WinSCP.
2. Enter 172.20.2.1 under **Host Name**.
3. Enter the credentials for host access to the Nerve Device below under **User name** and **Password**.



4. Navigate to the **/tmp/** directory on the right side of the WinSCP window. It is located in the **root** directory.



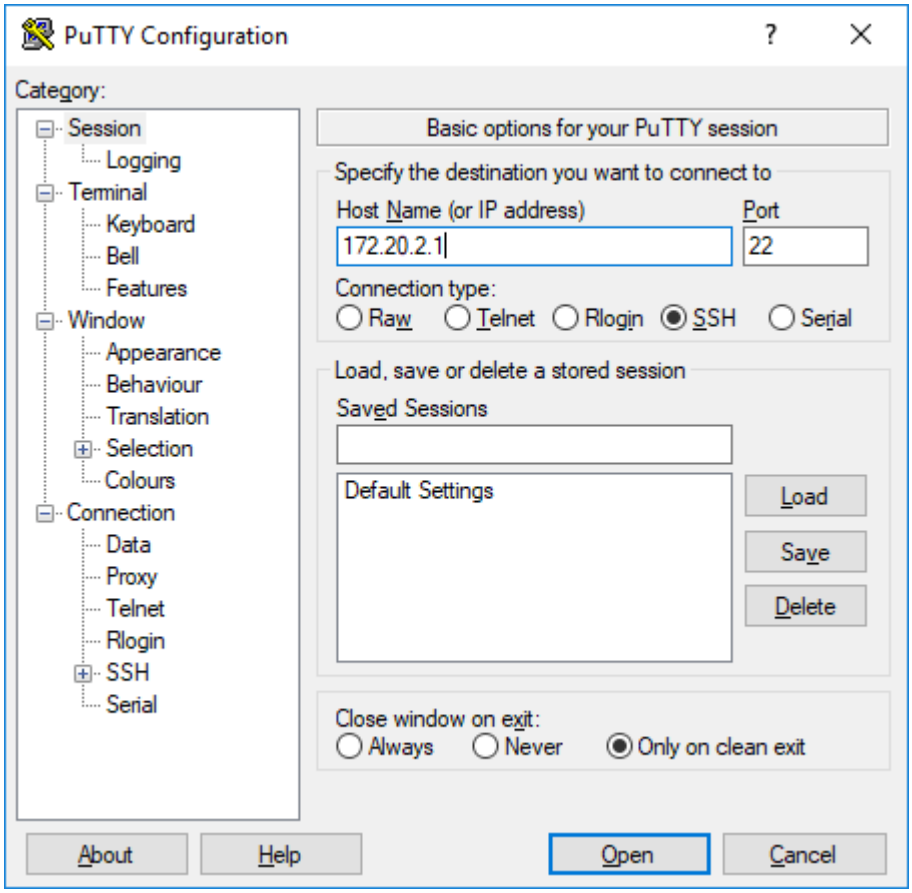
5. Copy the ISO file of the OS that is to be installed on the virtual machine to the **tmp** folder on the Nerve Device.

Creating a Virtual Machine on a Node

Using the Virtual Machine Manager is recommended to create a virtual machine and install the OS from the ISO file. Note that the virtual machine in this chapter is a fresh installation and will be generated on the MFN 100 directly.

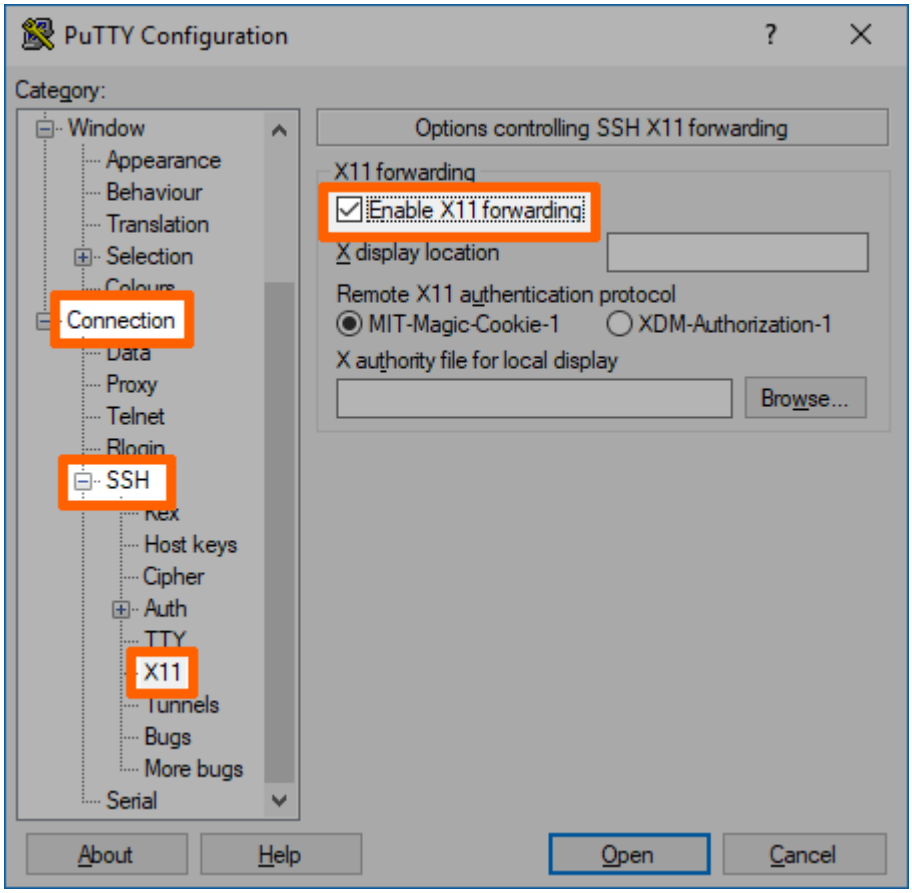
1. Run Xming or an alternative.
2. Open an SSH client like PuTTY.

3. Enter 172.20.2.1 under **Host Name (or IP address)** to log in to the host of the Nerve Device.



4. Expand **Connection > SSH > X11** on the left side.

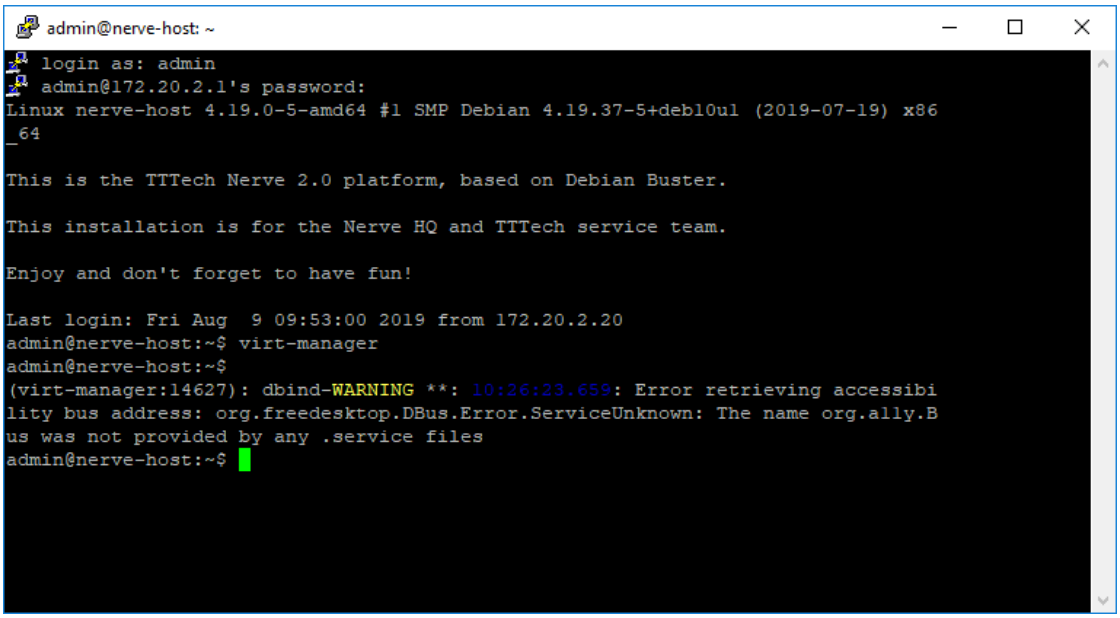
5. Tick the checkbox next to **Enable X11 forwarding**.



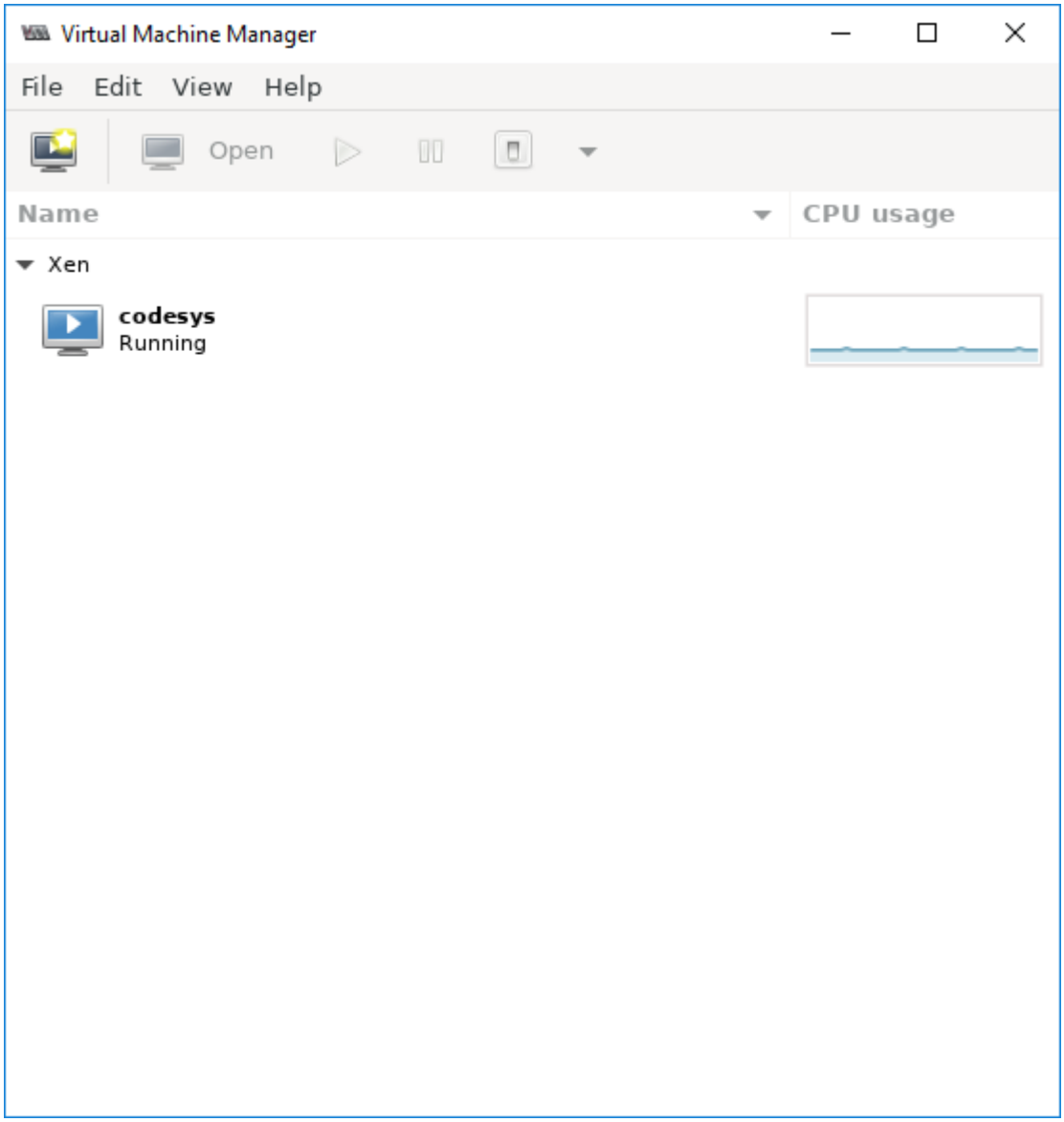
6. Click **Open**.

7. Log in with the credentials for host access of the Nerve Device.

8. Enter virt-manager.



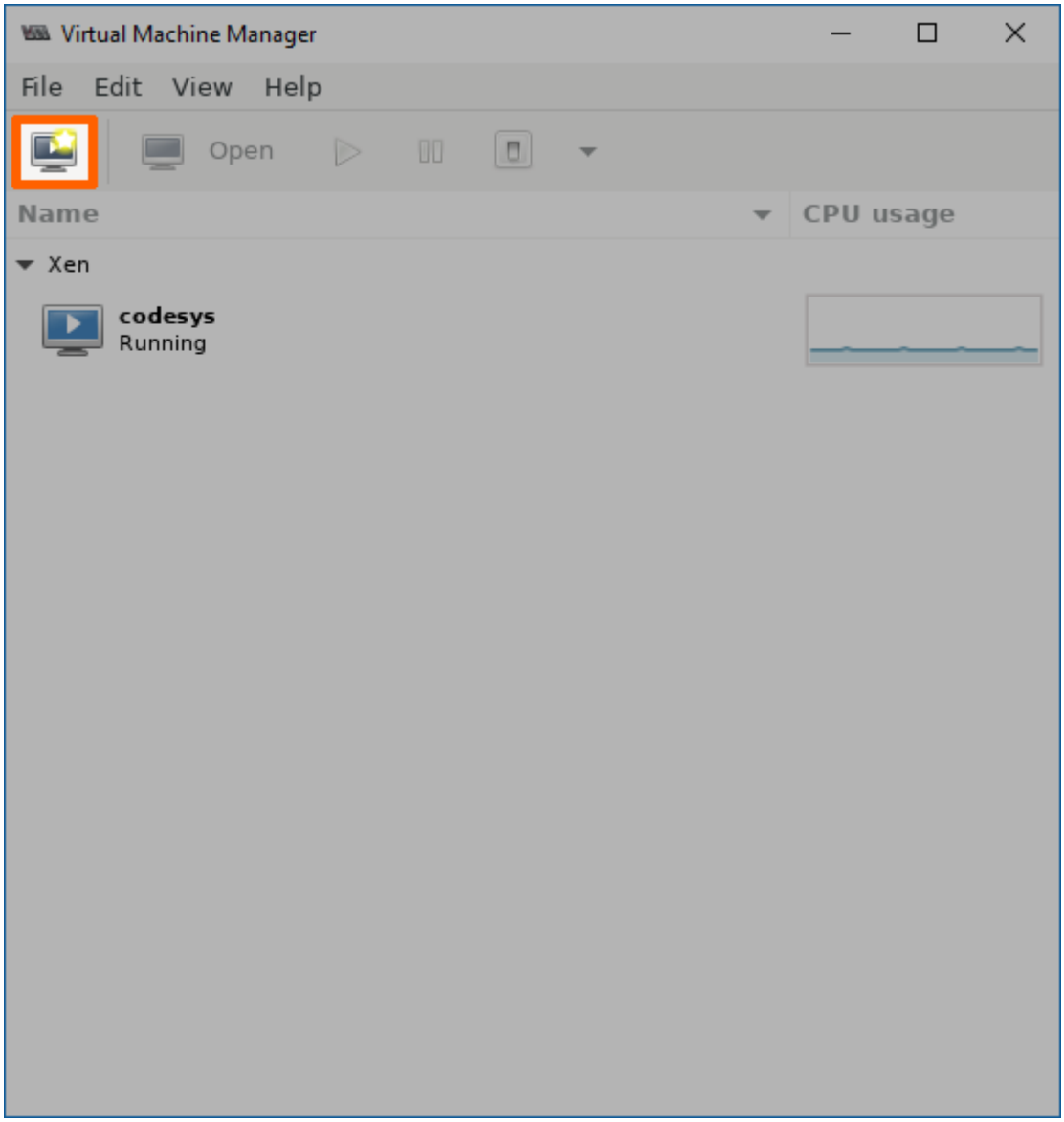
The interface of the Virtual Machine Manager will open.



Inserting the Installation File (ISO)

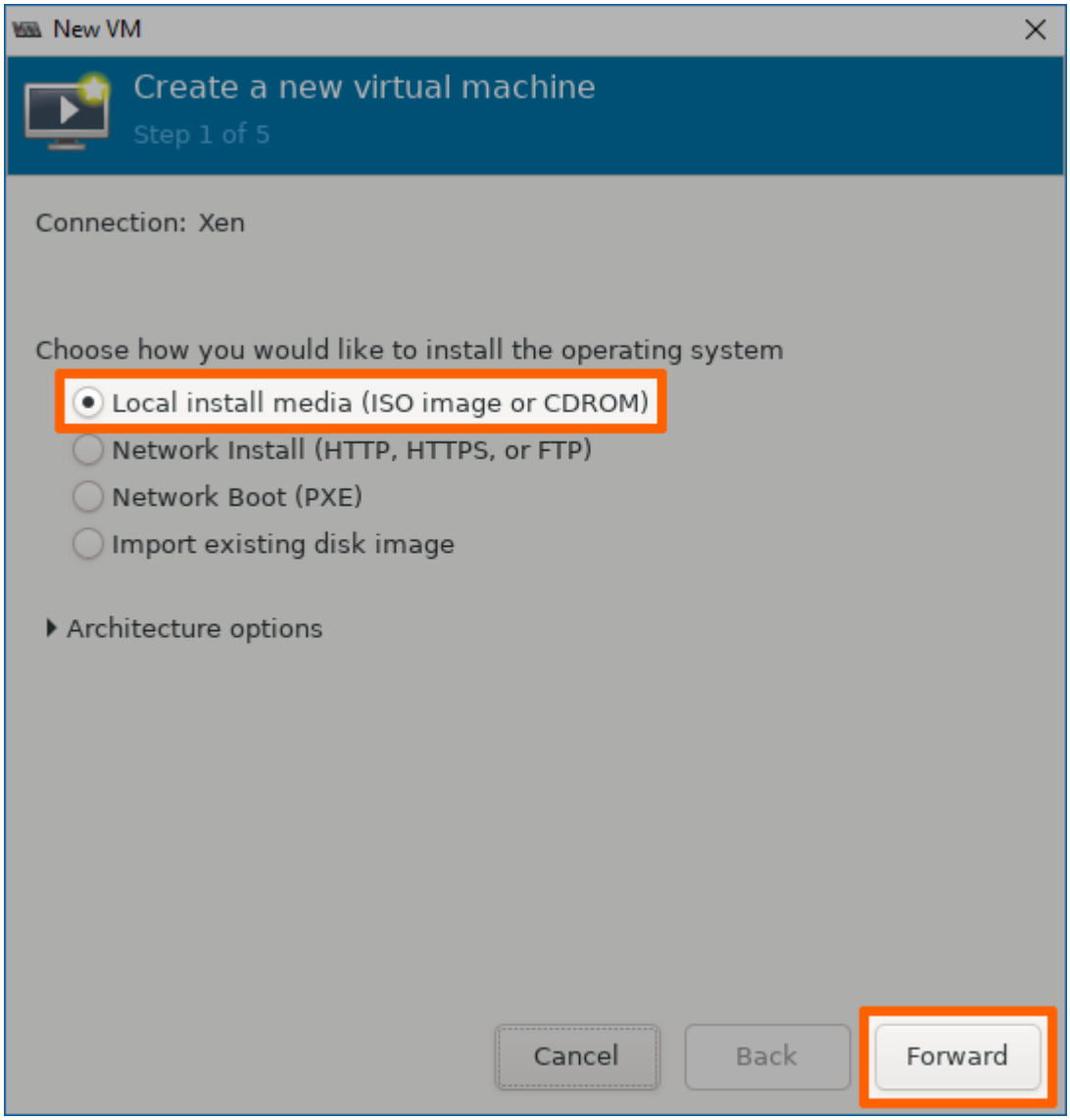
The creation of the virtual machine can now be initiated with the installation of the OS following right after. Note that the Virtual Machine Manager requires the virtual insertion of the ISO file in the beginning while resources for the virtual machine are defined later.

1. Select **File > New Virtual Machine** or click the symbol.



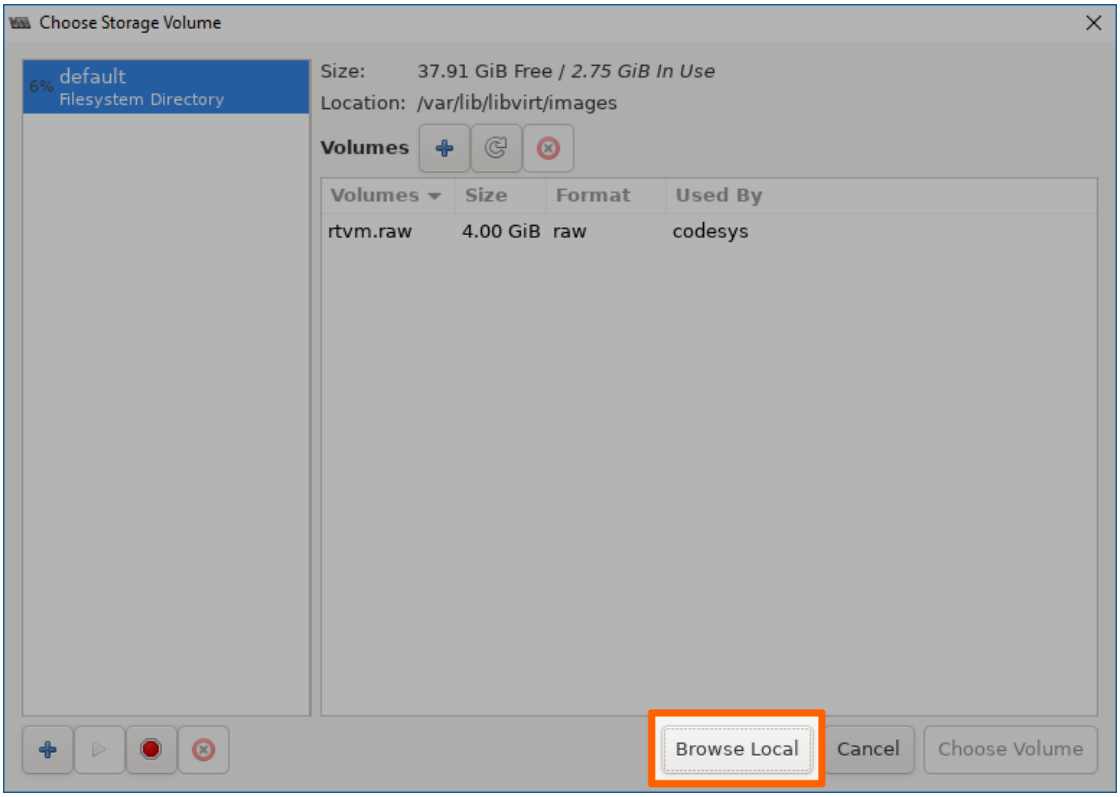
2. Select **Local install media (ISO image or CDROM)**.

3. Click **Forward**.



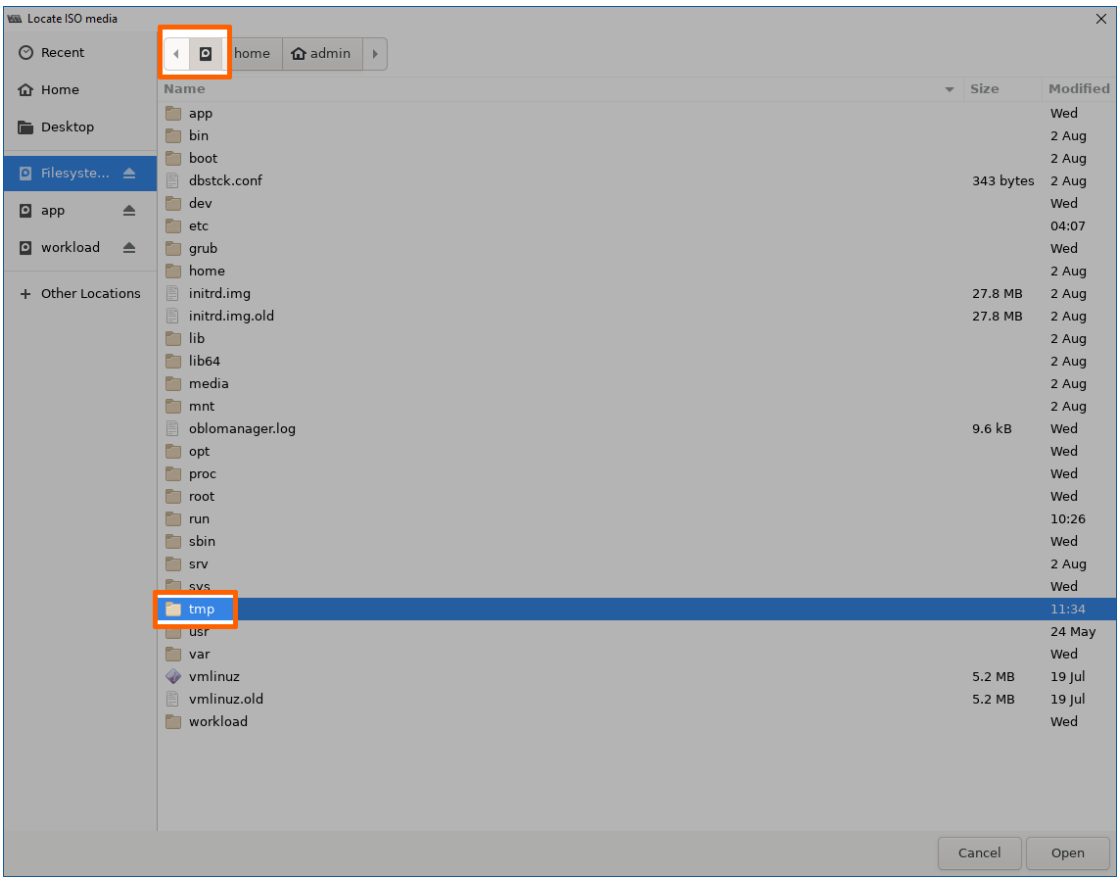
4. Click **Browse...** in the next window.

5. In the new window select **Browse Local**.



6. Navigate to the **tmp** folder by clicking the left arrow next to **admin**. It is located in the root directory.

7. Double-click the **tmp** folder.



Select the ISO file of the OS that you have copied before.

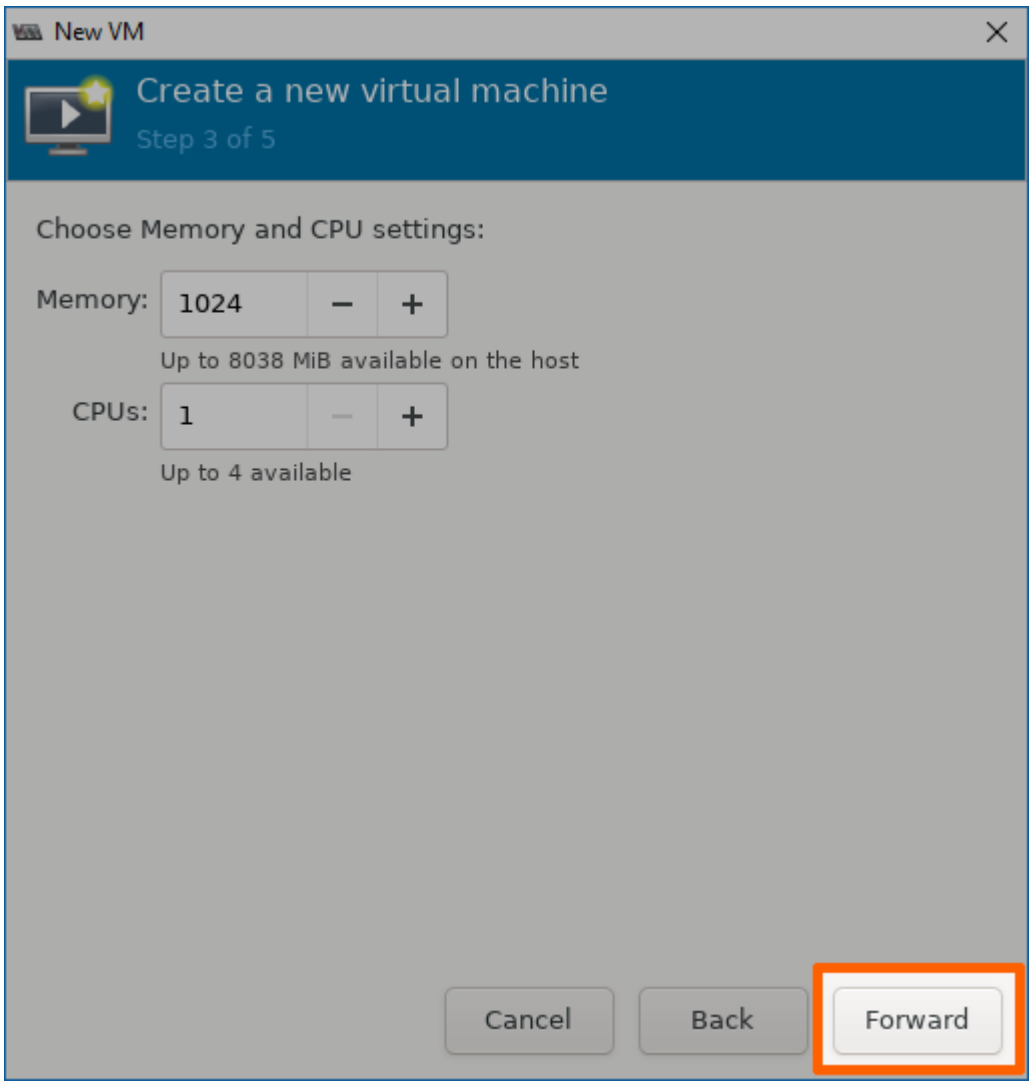
8.
9. Click **Open**.

Defining Resources for the Virtual Machine

Next, the amount of memory, the number of CPUs and the size of the logical volume assigned to the virtual machine need to be defined.

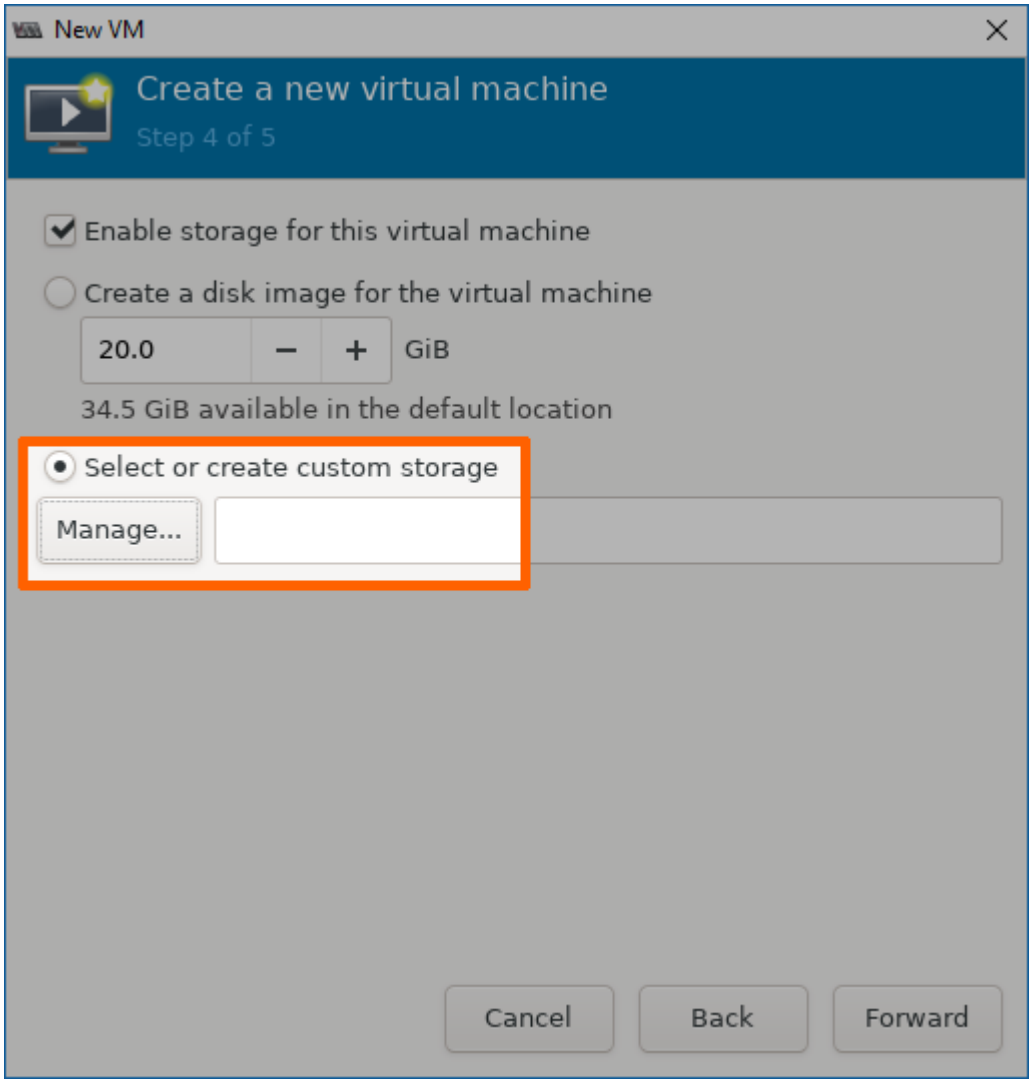
1. Click **Forward**.
2. Define how much memory and how many CPUs to assign to this virtual machine.

3. Select **Forward**.

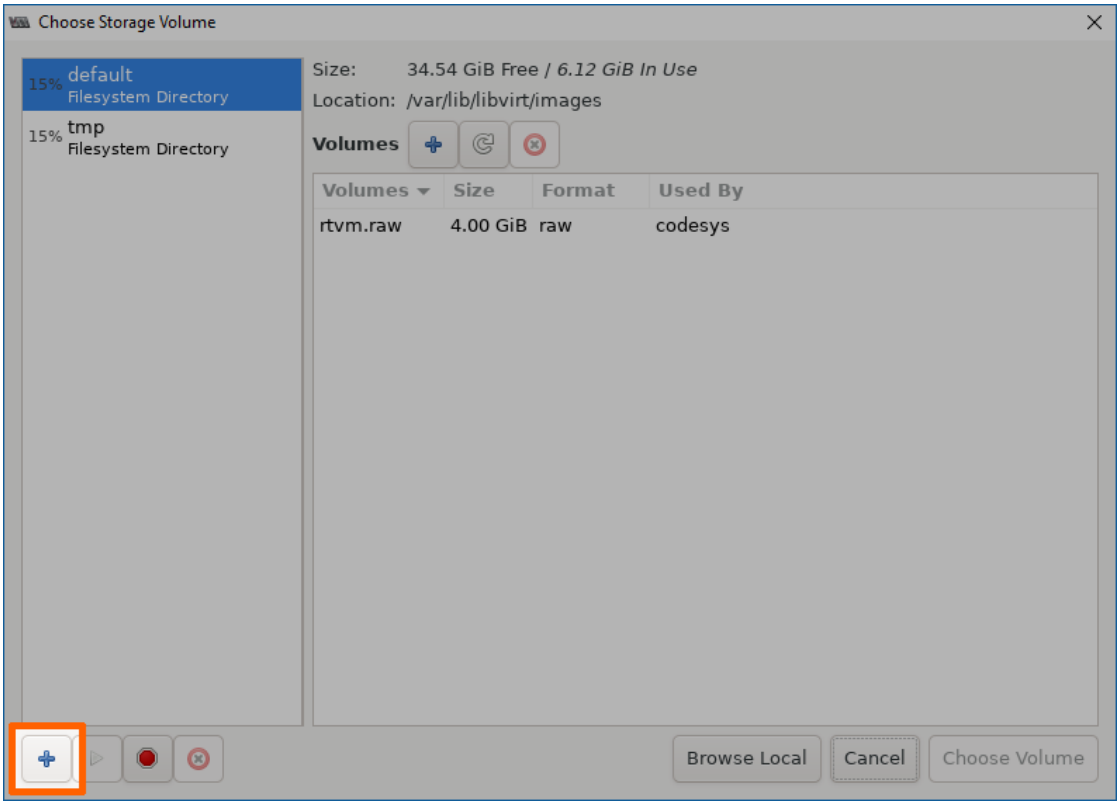


4. Click **Select or create custom storage**.

5. Select **Manage...**.



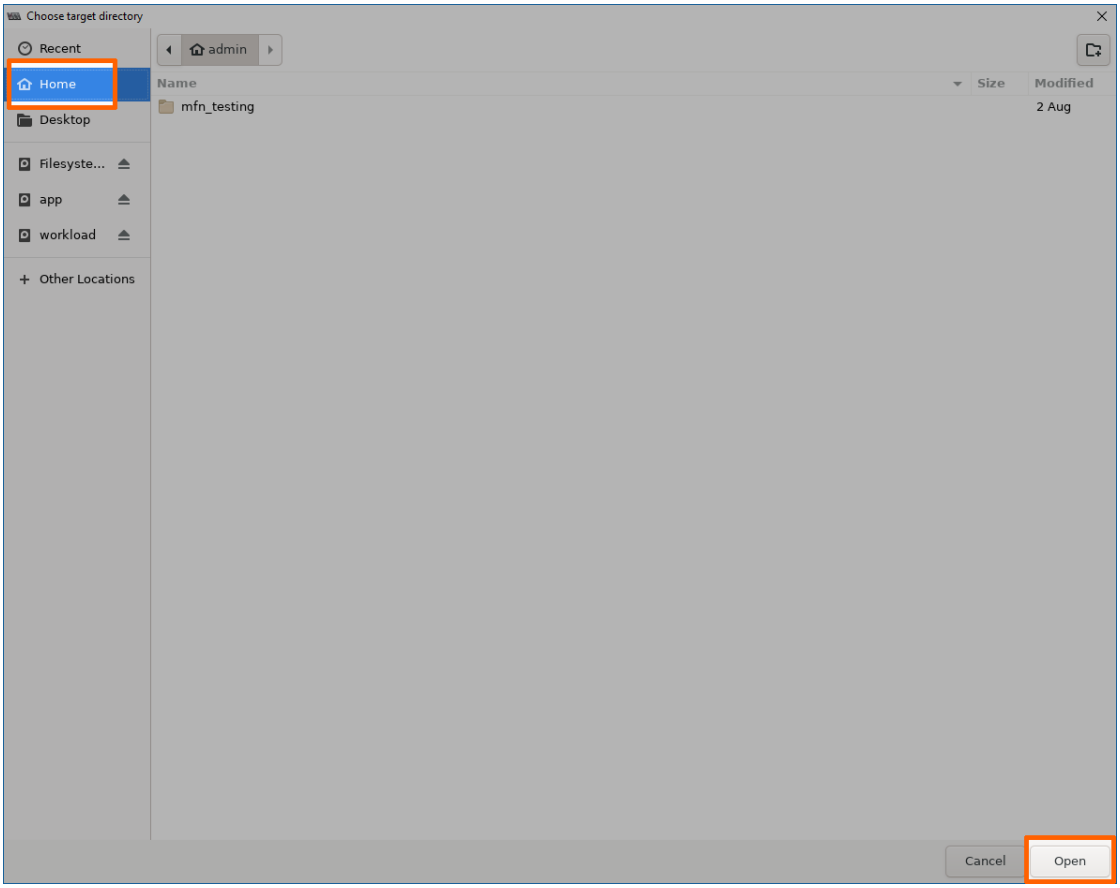
6. Click the **+** icon in the lower-left corner to add a new logical volume group (**Add Pool**).



7. Enter a name for the storage pool type.

8. Click **Forward**.
9. Click **Browse**.
10. Select **Home** on the left side.

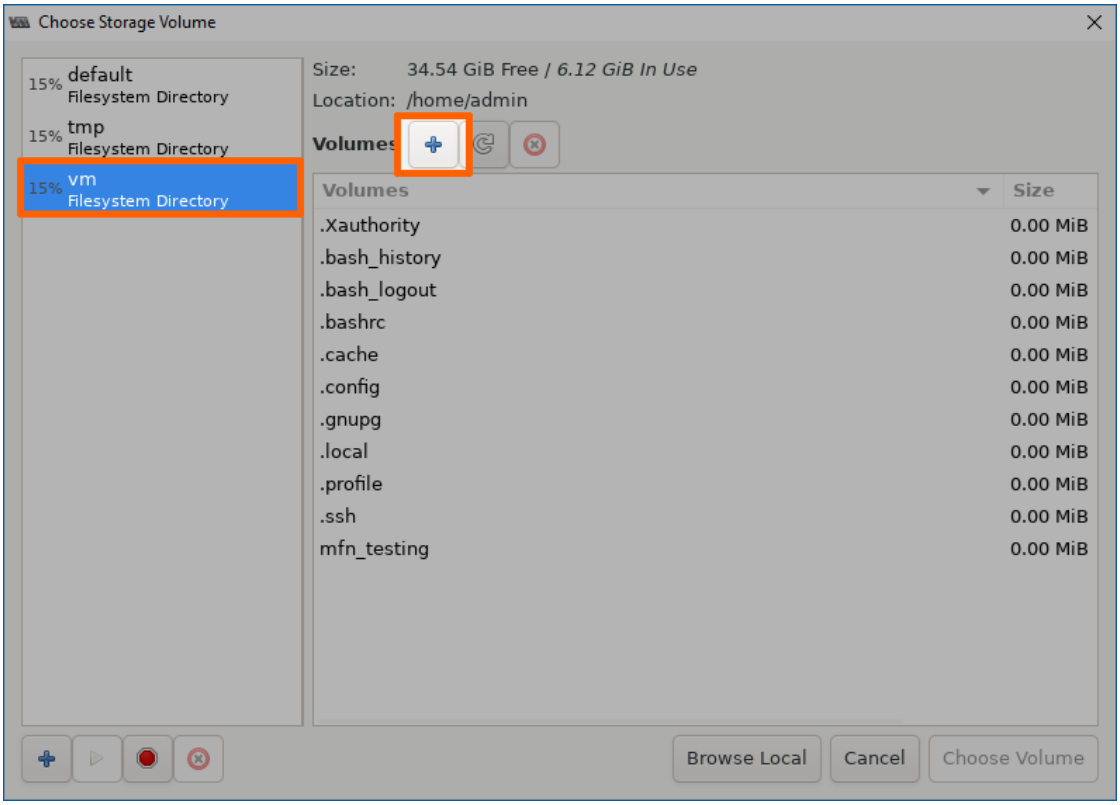
11. Click **Open** in the lower-right corner.



12. Select **Finish**.

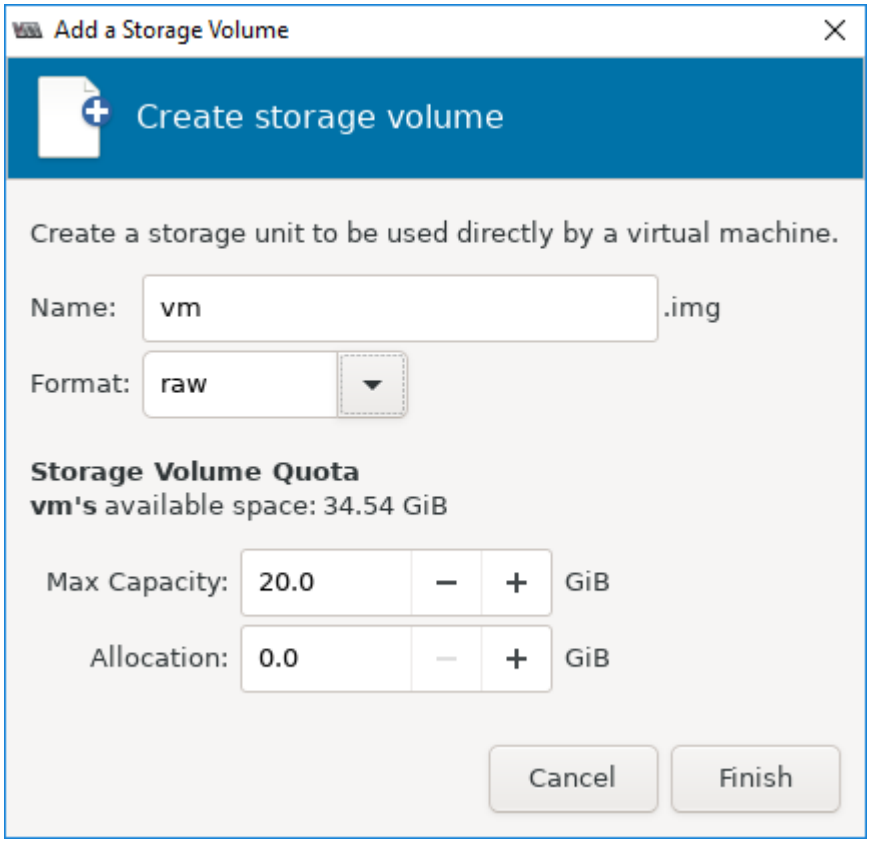
13. Select the new pool you have just created on the left side.

14. Click the **+** symbol next to **Volumes**.



15. Enter the following information in the new window:

Item	Description
Name	Enter a name for the storage volume used by the virtual machine.
Format	Select **raw** from the drop-down menu.
Max Capacity	Choose the storage capacity of the storage volume.

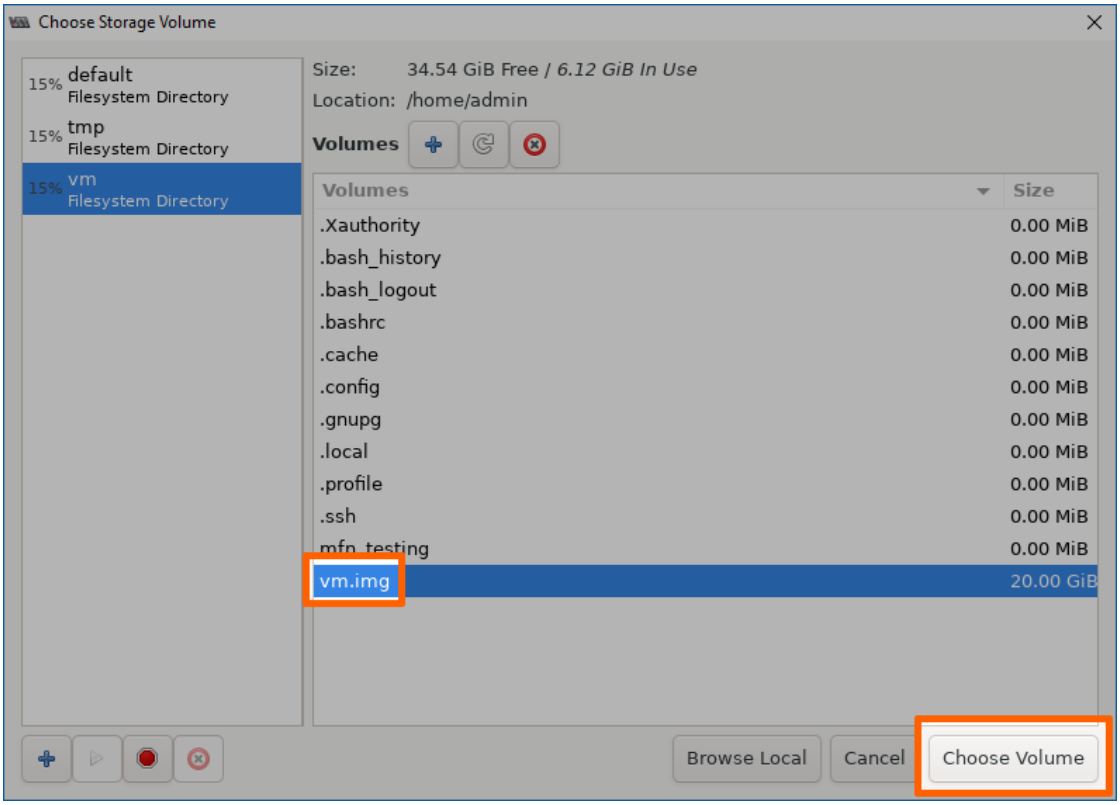


16. Click **Finish**.

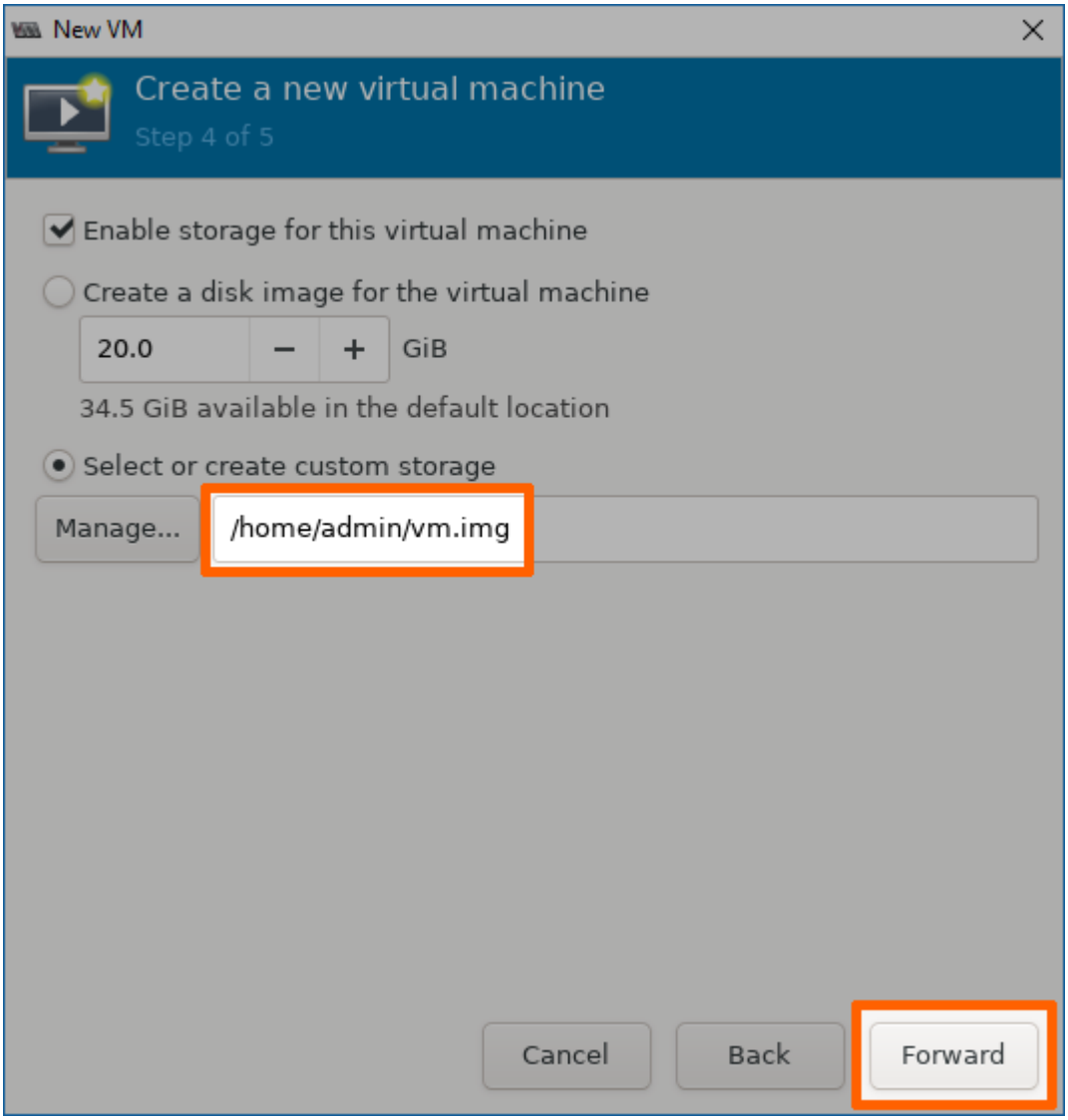
Installing the Operating System

Now the virtual machine will be initiated and the installation of the OS will be started.

1. Select the image that was created in the **Choose Storage Volume** window.
2. Click **Choose Volume**.

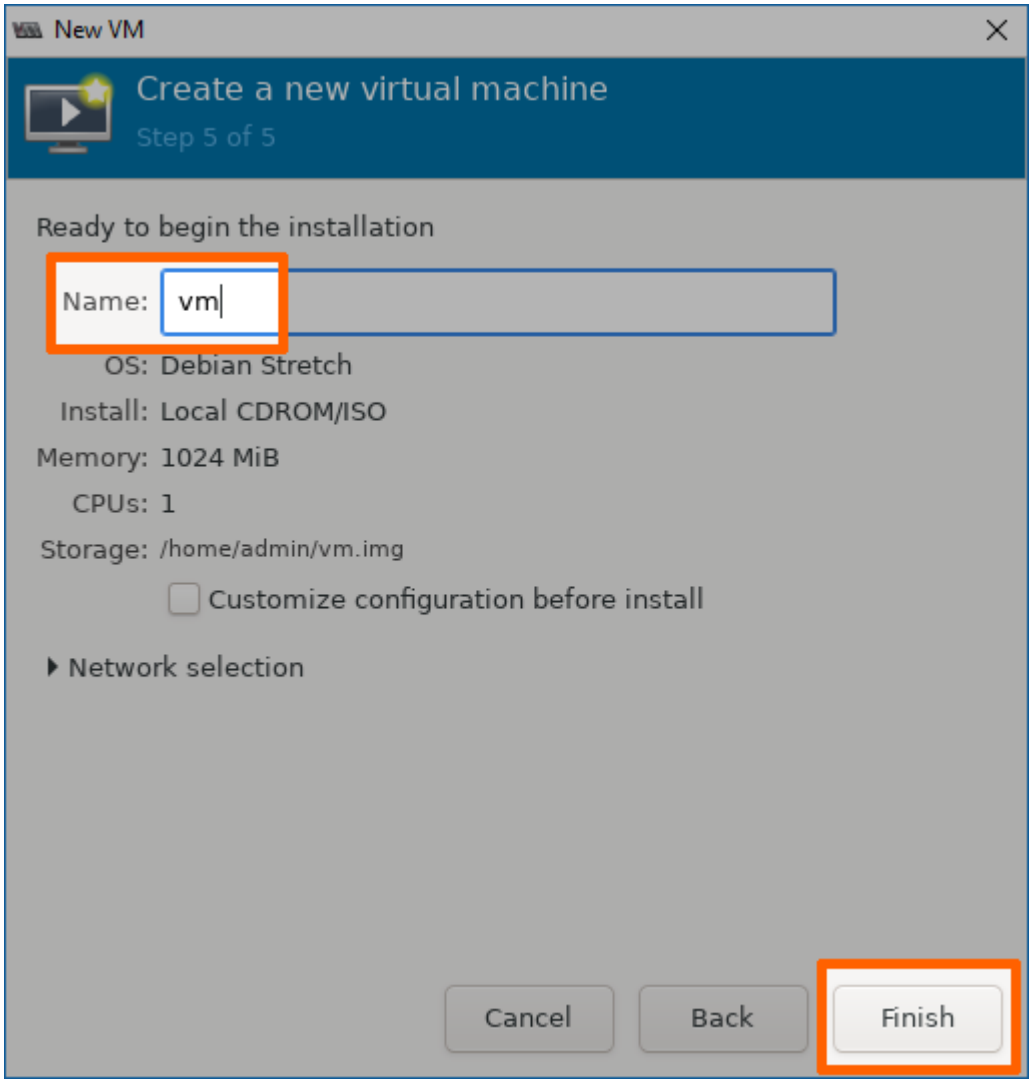


3. Click **Forward** to initialize the installation of the OS on the virtual machine.



4. Enter a name for the virtual machine.

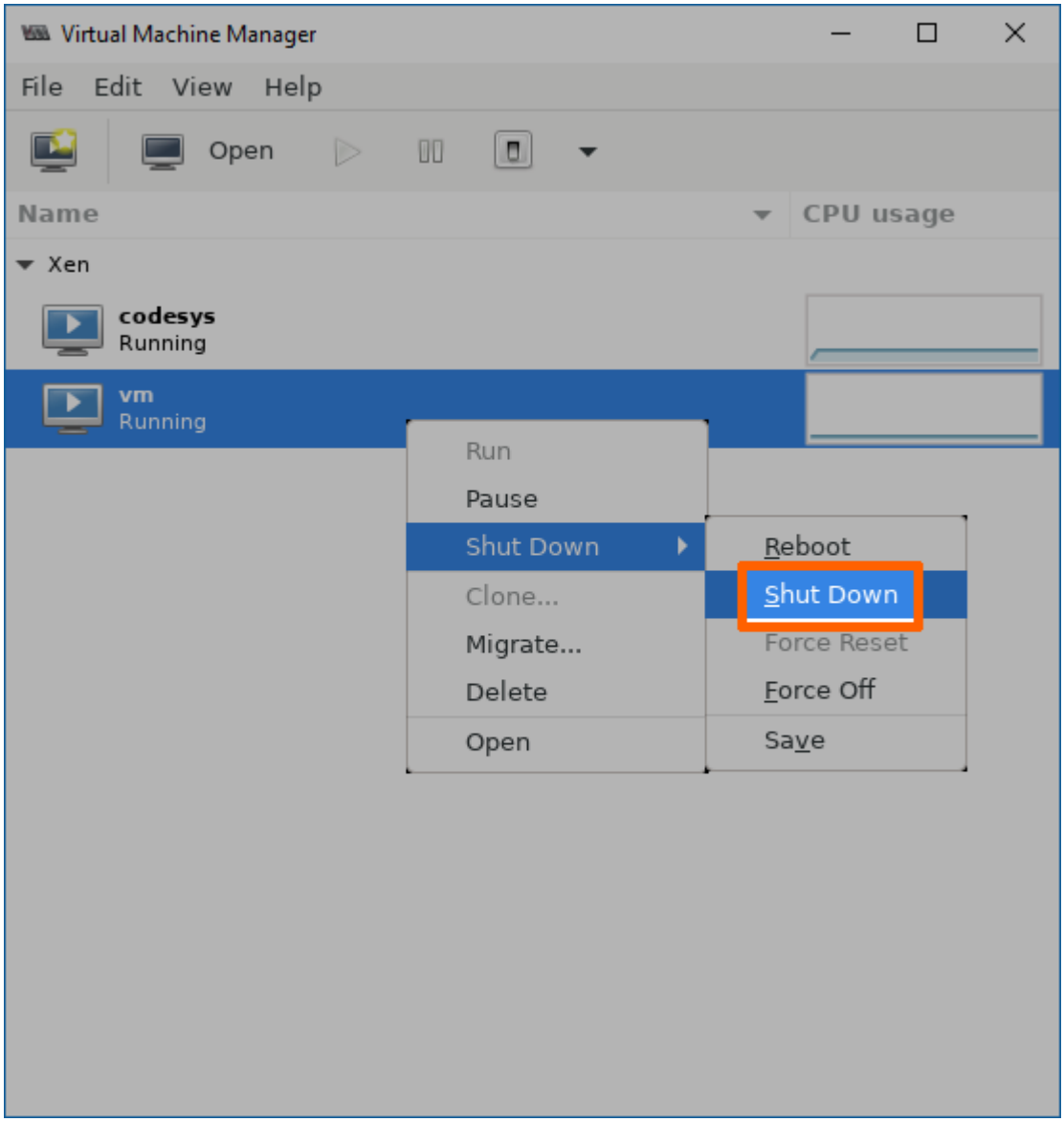
5. Click **Finish**. The virtual machine will be initiated and the installation of the OS will be started.



6. Complete the installation of the OS. Please follow the steps provided by the vendor.

7. After the installation is completed, right-click the virtual machine in the main Virtual Machine Manager window.

8. Select **Shut Down > Shut Down** to shut down the VM.



Copying the IMG File to a Local Workstation

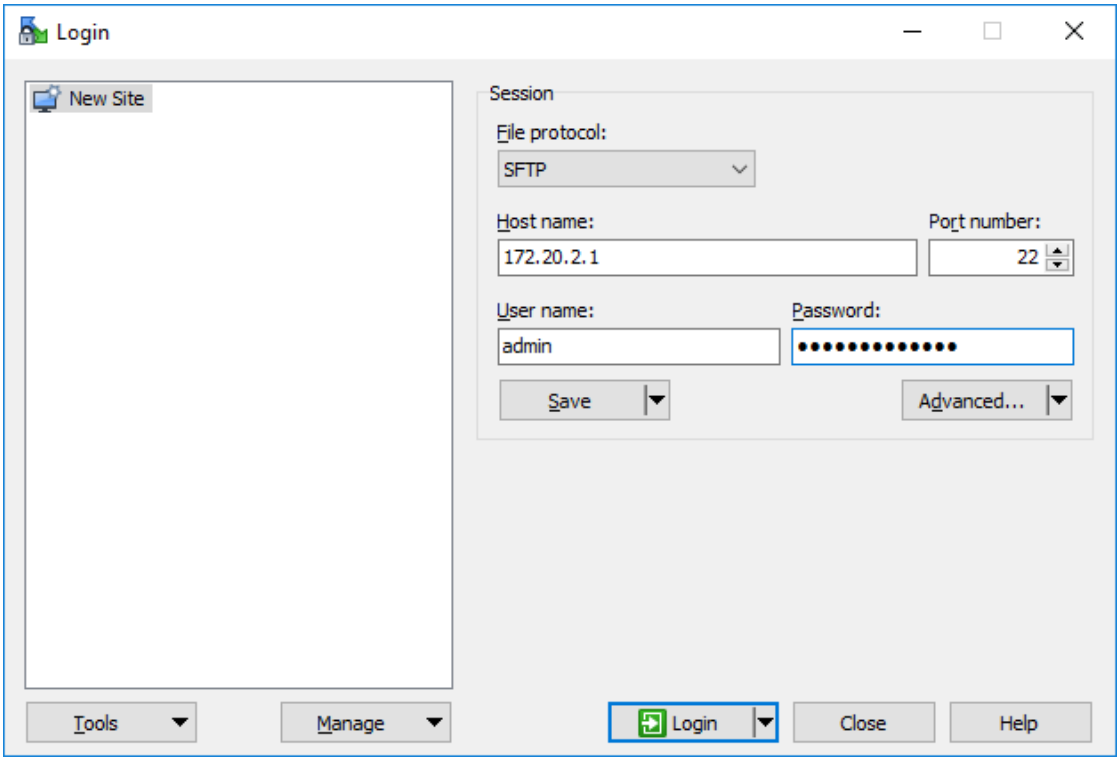
With the generation of the VM, the IMG file of the VM has also been generated on the Nerve Device. Now you can copy the IMG file to your local workstation.

1. Switch to the SSH client window.
2. Enter sudo chmod o+r /home/admin/<yourvm>.img.

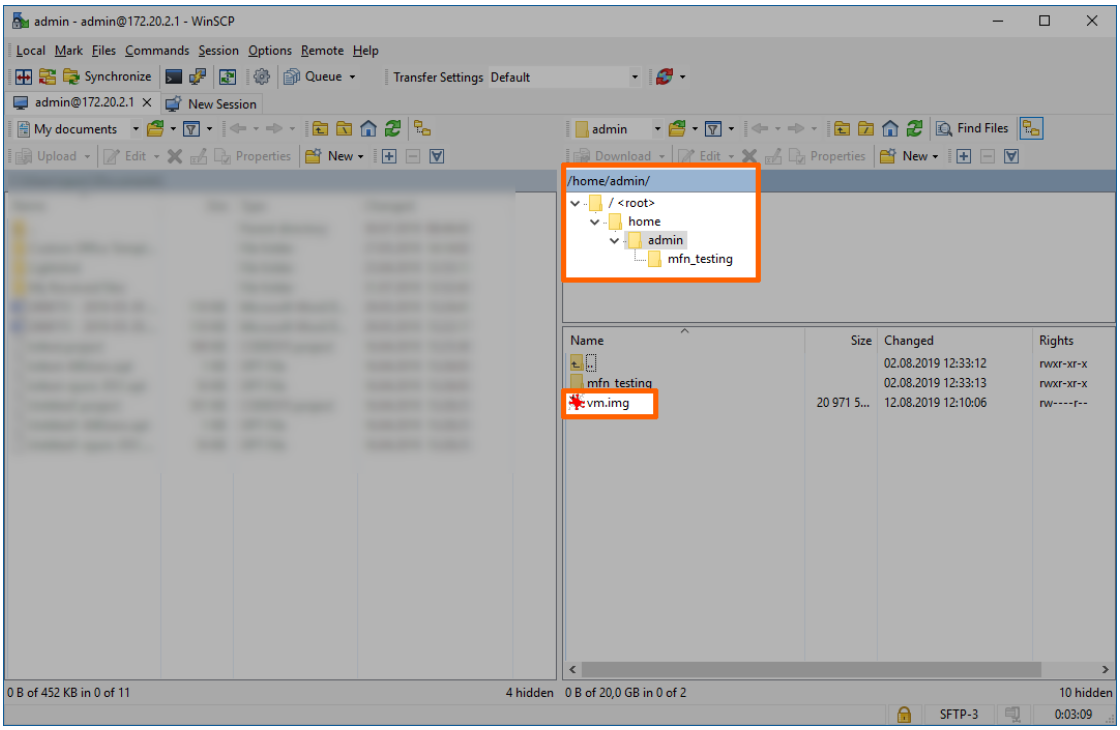
> **NOTE**
>
> The IMG file is called vm.img in this example. Please replace the placeholder name of the image in the command with the name of your IMG file.

3. Open a file transfer client like WinSCP.
4. Enter 172.20.2.1 under **Host Name**.

5. Enter the credentials for host access to the Nerve Device below under **User name** and **Password**.



6. Copy <yourvm>.img to your local workstation.



The virtual machine has now been generated on the Nerve Device and the IMG file of the virtual machine is now on your local workstation.

NOTE

Do not deploy the virtual machine from the process above to the same Nerve Device through the Management System. The virtual machine will be present twice. The deployment of the virtual machine from the process above should be done to different nodes.

Obtaining the XML file

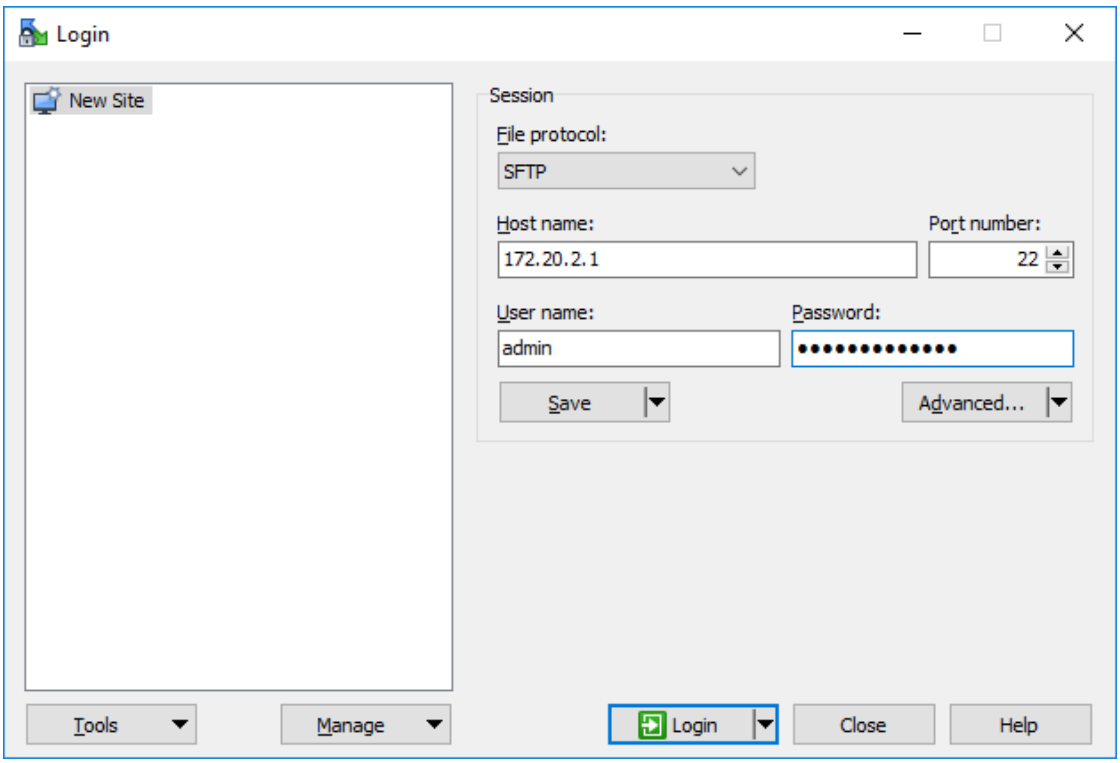
When the IMG was generated on the Nerve Device, an XML file for the IMG file was generated as well. It also has to be obtained manually.

1. Switch to the SSH client window.
2. Enter sudo chmod o+r /etc/libvirt/libxl/<yourvm>.xml.

NOTE

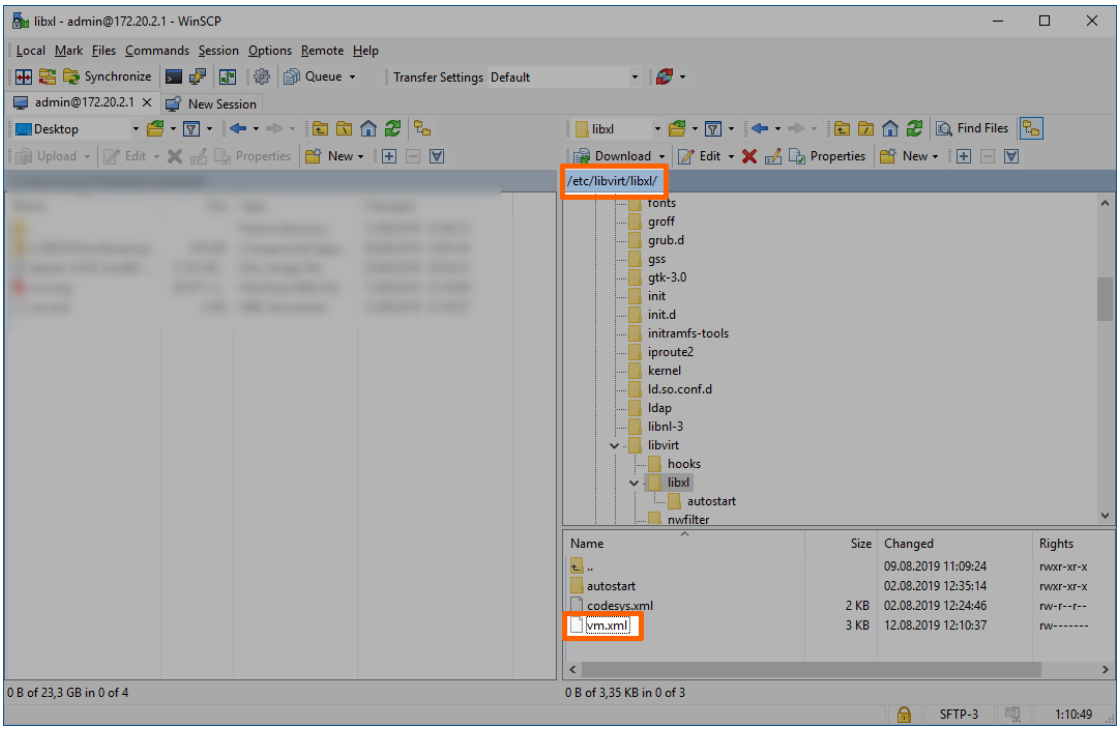
The XML file is called vm.xml in the screenshots of this example. Please replace the placeholder name of the XML file in the command with the name of your XML file.

3. Open a file transfer client like WinSCP.
4. Enter 172.20.2.1 under **Host Name**.
5. Enter the credentials for host access to the Nerve Device below under **User name** and **Password**.



6. Change the path to /etc/libvirt/libxl/ on the Nerve Device. You can find the **etc** directory in the **root** directory.

7. Copy the <yourvm>.xml file to your local workstation.



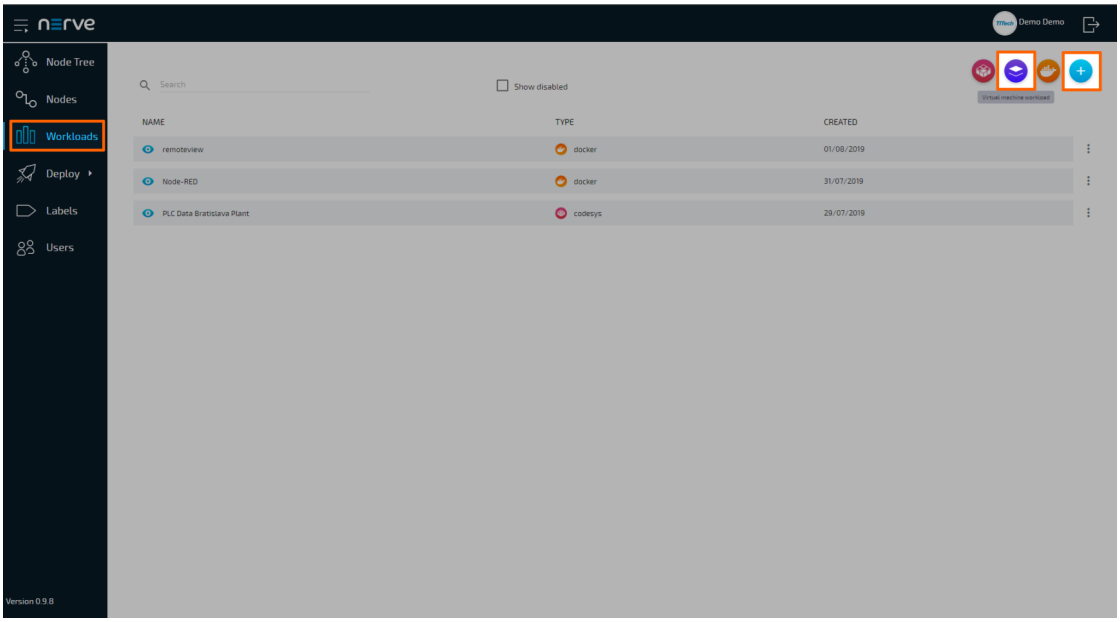
With this you have all the necessary files to provision a Virtual Machine workload for this virtual machine.

Provisioning a Virtual Machine Workload

The following instructions cover the basic requirements for provisioning a Virtual Machine workload. Optional settings will be left out. Extended options are addressed in the last section of this chapter.
There are two further types of workloads that can be provisioned: CODESYS workloads and Docker workloads. The process for each workload is highlighted in its respective chapter.

1. Log in to the Management System.
2. Select **Workloads** in the left-hand menu.
3. Select **+** in the upper-right corner.

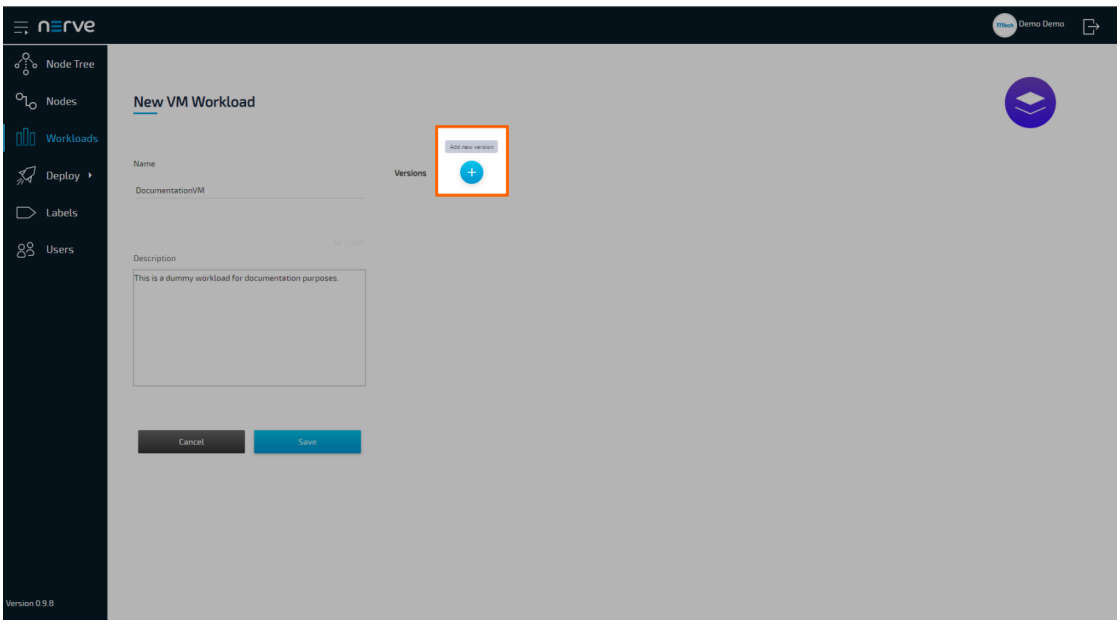
4. Select the virtual machine symbol (**Virtual machine workload**) in the middle of the three symbols that expanded.



5. In the new window, enter a name for your workload.

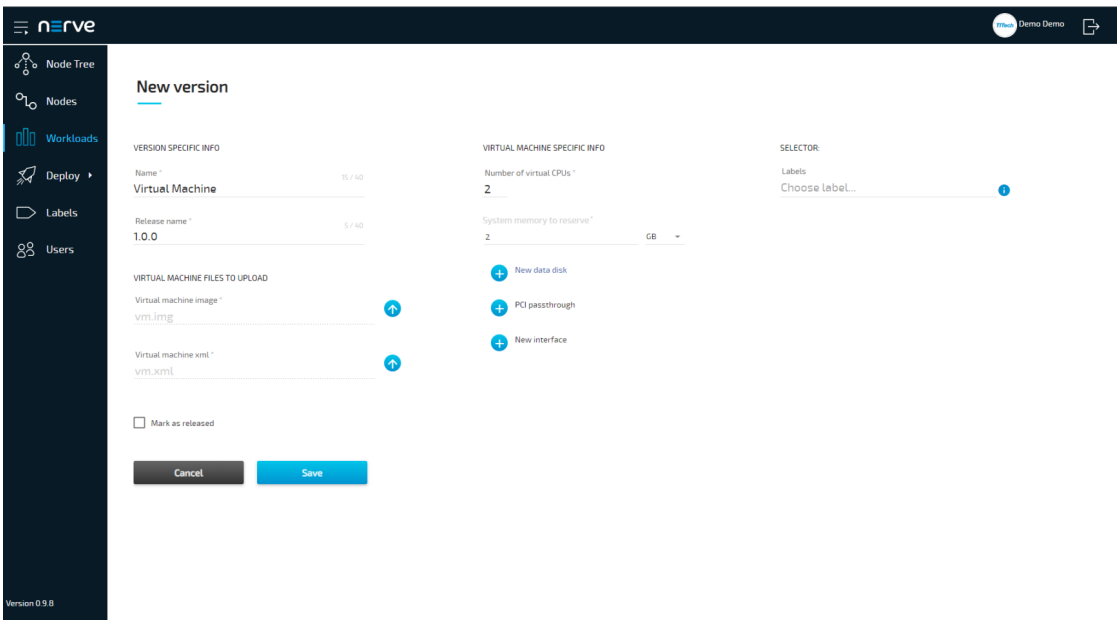
> **NOTE**
>
> Do not use codesys as a name for a Virtual Machine workload. Deploying a VM workload named codesys will interfere with the system.

6. Select **+** next to **Versions** to add a new version of the workload.



7. In the next window, enter the following information:

Item	description
Name	In the new window, enter a **Name** for the version of this workload.
Release name	Enter a **Release name** for the version of this workload.
Virtual machine image	Click the **upward arrow** symbol here to add the virtual machine image. The image has to be an IMG or RAW file.
Virtual machine xml	Click the **upward arrow** symbol here to add the virtual machine XML file.
Number of virtual CPUs	Enter the number of virtual CPUs you would like to use for this virtual machine.
System memory to reserve	Assign how much system memory you would like to reserve for this workload.



8. Select **Save** in the lower-left corner.

The workload has now been provisioned and is ready to be deployed in the **Deploy** menu.

> **NOTE**
>
> While some settings are not required to provision a Virtual Machine workload in the Management System, you will have to fill out additional settings for the workload to perform as desired. Depending on the virtual machine you want to deploy, you may have to define new interfaces. Please keep this in mind and make sure to learn the details about your virtual machine.

Settings for Virtual Machine Workloads

In the instructions above, all optional settings have been left out. Below is an overview of all the options with an explanation to each option.

Setting	Description
VERSION SPECIFIC INFO	**Name** A name for your workload version. This could be a reminder for a certain configuration. Example: "Unlimited" as a name for a virtual machine that has unlimited access to CPU resources. **Release name** A release name for your workload version. This could be a version number. Example: 1.0.1
VIRTUAL MACHINE FILES TO UPLOAD	Two files need to be added here: **Virtual machine image** Upload the virtual machine image with the file extension RAW or IMG here. Do this by clicking the **upward arrow** symbol and selecting the file in the file browser. This is the first of the two files you generated in the process before. **Virtual machine xml** Upload the virtual machine XML file here. Do this by clicking the **upward arrow** symbol and selecting the file in the file browser. This is the second of the two files you generated in the process before. Please be aware that the settings defined under **Virtual machine specific info** are going to overwrite parts of this XML file.

Setting	Description
Virtual machine specific info	**Number of virtual CPUs** Define the number of virtual CPUs you would like to assign to this virtual machine. The CPUs are then reserved exclusively for the Virtual Machine workload and cannot be used by other processes. This setting is mandatory and the workload cannot be provisioned if it is left blank. **System memory to reserve** Assign how much system memory you would like to reserve for this workload. The memory assigned here will be reserved exclusively for this Virtual Machine workload and will not be available for any other processes. This setting is mandatory and the workload cannot be provisioned if it is left blank. **Add new data disk** Click here to add a new data disk for the virtual machine. This data disk functions like an extra hard drive for data separate from the virtual machine. Enter a **Data disk name** and define the **Disk size**. **Add PCI passthrough** Click here to add a PCI passthrough to the virtual machine. Enter the PCI address of the interface you would like to pass through to be directly used by the virtual machine. Please note that the PCIe address is specific to a certain hardware. If you use this option, you should probably limit the installation targets to Nodes with this specific hardware by using selectors. See the user guide for more information on selectors. **Add new interface** Click here to add a new interface. You can choose between a bridged interface and a NAT-interface. For NAT interfaces you can define port mappings for TCP and UDP. The names of the interfaces here have to match the names of the pre-defined network interfaces. Please refer to the networking chapter in the user guide for more detailed information.
SELECTOR	**Labels** If you have defined labels and assigned them to nodes, you can add them as selectors to the workload. When deploying a workload, the list of nodes will be filtered automatically to the specified label.
Mark as released	Tick this checkbox if you want to mark this workload as released. Once marked as released, the workload cannot be edited anymore.

Working with Docker Containers

In order to work with Docker containers on nodes, workloads need to be provisioned in the Management System. Here, provisioning is the creation of a workload and its storage in the workload repository of the Management System so that it can be deployed to nodes. This requires

configuration of the workload and files that need to be uploaded to the Management System. After that, the workload can be deployed to nodes.

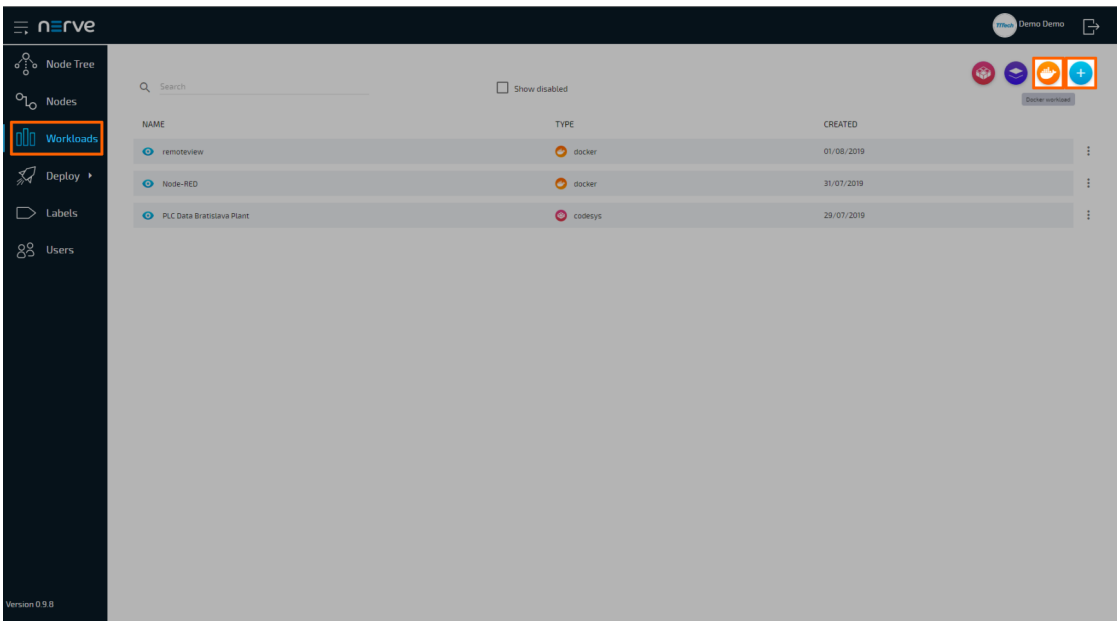
The following instructions cover the basic requirements for provisioning a Docker workload. Optional settings will be left out. Extended options are addressed in the last section of this chapter.
There are two further types of workloads that can be provisioned: CODESYS workloads and Virtual Machine workloads. The process for each workload is highlighted in its respective chapter.

NOTE

A Docker image is required for the following instructions. Consult the Docker documentation for help on creating a Docker image.

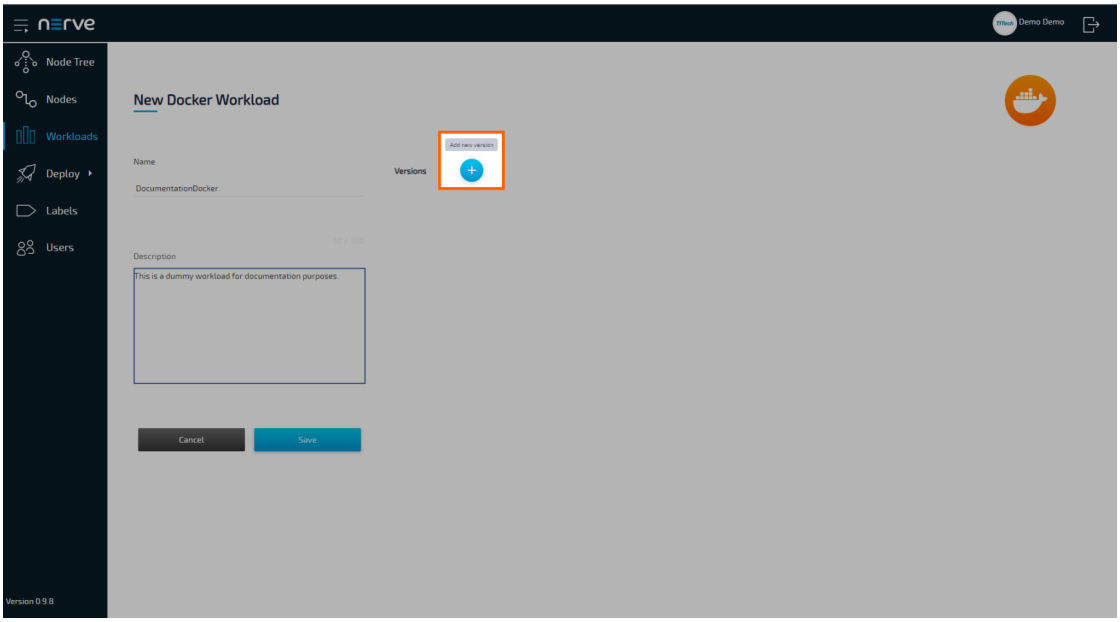
1. Log in to the Management System.
2. Select **Workloads** in the left-hand menu.
3. Select **+** in the upper-right corner.

4. Select the Docker symbol (**Docker workload**) on the right of the three symbols that expanded.



5. In the new window, enter a name for your workload.

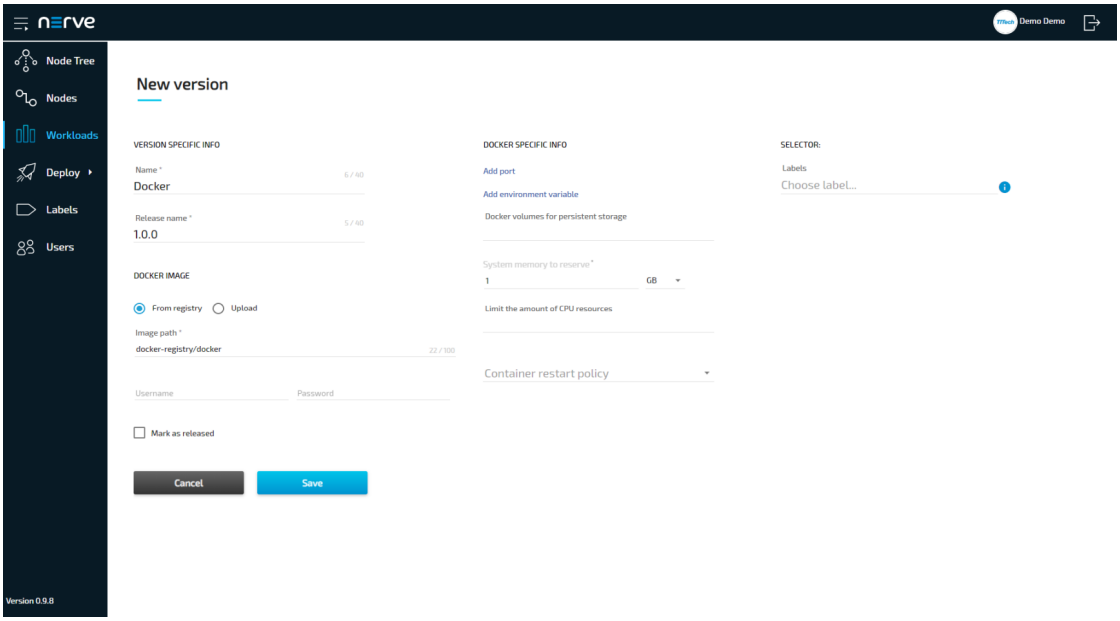
6. Select **+** next to **Versions** to add a new version of the workload.



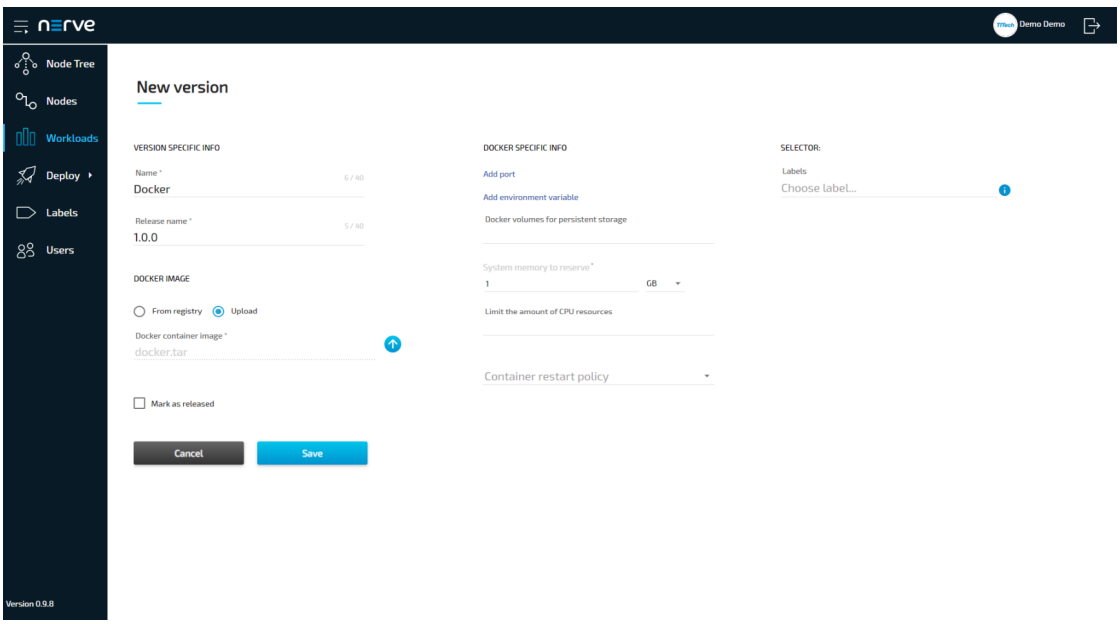
7. In the next window, enter the following information:

Item	Description
Name	Enter a **Name** for the version of this workload.
Release name	Enter a **Release name** for the version of this workload.
DOCKER IMAGE	Select **From registry** or **Upload**. If you select **From registry**, you need to enter the Docker registry to the Docker container image under **Image path**. If you select **Upload**, click the **upward arrow** symbol to upload the Docker container image.
System memory to reserve	Assign how much system memory you would like to reserve for this workload.

From registry:



Upload:



8. Select **Save** in the lower-left corner.

The workload has now been provisioned and is ready to be deployed in the **Deploy** menu.

> **NOTE**
>
> While some settings are not required to provision a Docker workload in the Management System, you will have to fill out additional settings for the workload to perform as desired. Depending on the Docker container you want to deploy, you will need to define ports and configure environment variables. Please keep this in mind and make sure to learn the details about your Docker container.

Settings for Docker Workloads

In the instructions above, all optional settings have been left out. Below is an overview of all the options with an explanation to each option.

Setting	Description
VERSION SPECIFIC INFO	**Name** A name for your workload version. This could be a reminder for a certain configuration. Example: "Unlimited" as a name for a Node-RED version that has unlimited access to CPU resources. **Release name** A release name for your workload version. This could be a version number. Example: 1.0.1
DOCKER CONTAINER FILE MANAGEMENT	You can select between two options here and either use a Docker registry URL to link to an online repository or upload the Docker container image from your workstation. **From registry** If you select this, you have to specify a URL pointing to the Docker container image under **Image path** below. Please note the differences between public Docker Hub registries and private registries. Private registries require the full URL to be specified, as well as a username and password if they require authentication. Public Docker Hub registries can be specified in their short form. Examples: • Public Docker Hub registry nodered/node-red-docker • Private registry with authentication and a tag at the end auth.docker.test.host.cloud/workload:v1.3 **Upload** If you select this, you have to upload the Docker container image from your workstation. Do so by clicking the **upward arrow** symbol to open your file browser.

Setting	Description
DOCKER CONTAINER SPECIFIC INFO	**Add port** Click here to define a **Host port** and a **Container port**. Please make sure to select the appropriate container port as it depends on the Docker container you are deploying. **Add environment variable** Click here to add an environment variable (**Env. variable**) and its **Variable value**. Please make sure to define the appropriate variables and values as they depend on the Docker container you are deploying, **Docker volumes for persistent storage** If your workload needs persistent storage, specify the path of the Docker volumes in the Docker container here. The volume for persistent storage is not erased even if the Docker workload is undeployed. **System memory to reserve** Assign how much system memory you would like to reserve for this workload. The memory assigned here is an upper limit that the Docker workload can use and is not exclusively reserved for the Docker workload. Other processes can use these resources as well. **Limit the amount of CPU resources** Specify here how many CPU cores the workload is allowed to use. If this field is left blank, the workload is allowed to use all available resources. **Container restart policy** Choose the container restart policy here to determine when the Docker container can be restarted. • **no** The container does not restart automatically. • **on-failure** The container restarts when it exits due to an error. • **always** The container restarts every time it stops. However, manually stopping the container is the exception. If a container is manually stopped, it is only restarted when the Docker daemon restarts or the container is restarted manually. • **unless-stopped** The container only restarts if it is manually stopped.
SELECTOR	**Labels** If you have defined labels and assigned them to nodes, you can add them as selectors to the workload. When deploying a workload, the list of nodes will be filtered automatically to the specified label.

Setting	Description
Mark as released	Tick this checkbox if you want to mark this workload as released. Once marked as released, the workload cannot be edited anymore.

Deploying a Workload

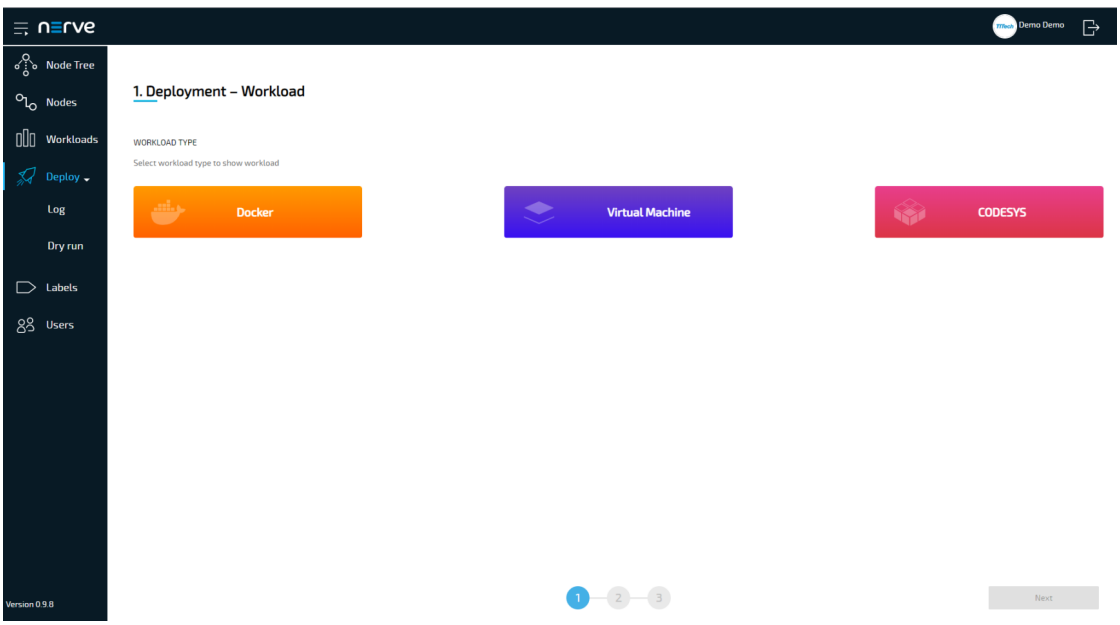
This chapter will take you through the deployment process of the available workloads: CODESYS workloads, Virtual Machine workloads and Docker workloads. The process of deploying workloads is identical for all three types of workloads. Therefore, the instructions below contain no specific information.
An example of the deployment of a specific workload follows in the next chapter: Workload Example: Installing Node-RED.

NOTE

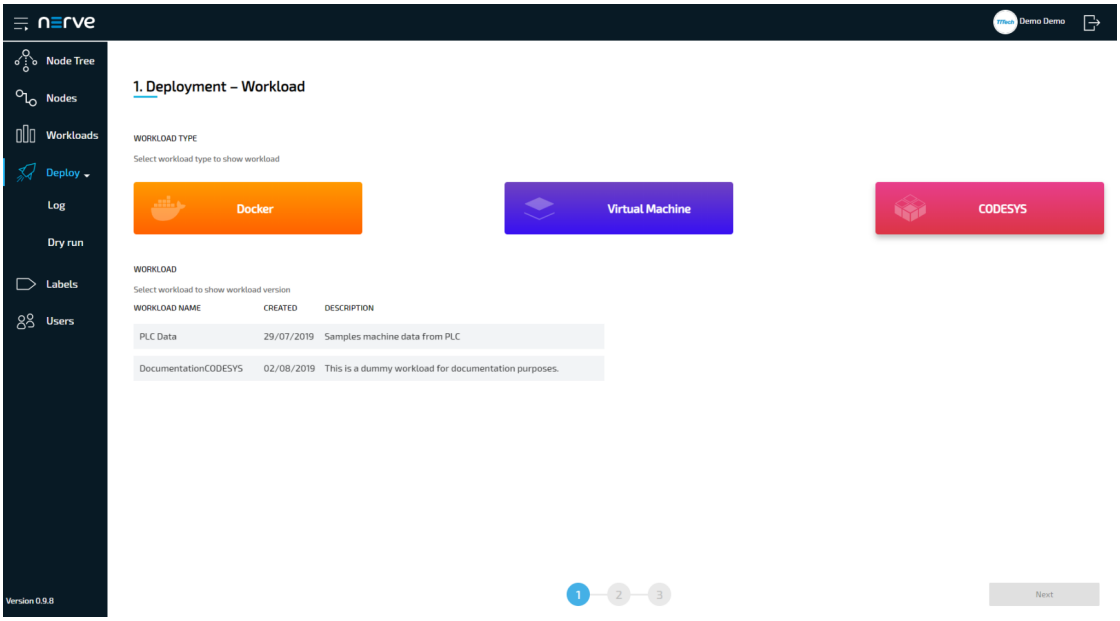
This chapter guides you through deploying a workload. If you want to test a deployment first, select **Deploy > Dry run** in the left-hand menu and follow the steps below starting from step 3.

However, please note that a successful dry run does not guarantee a successful deployment as it is only a simulation.

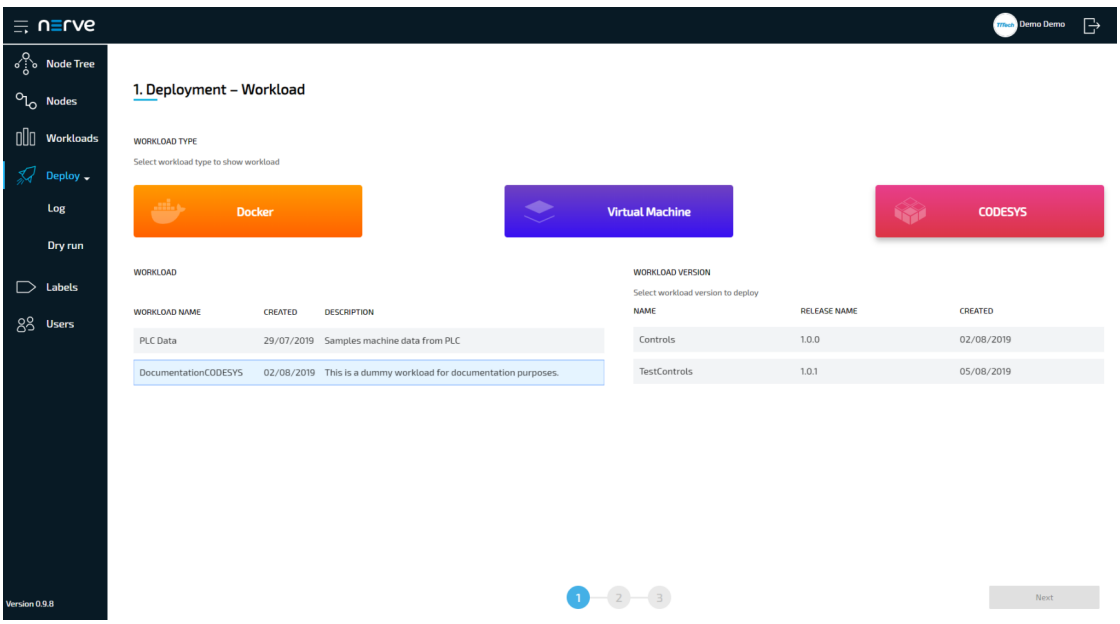
1. Select **Deploy** in the left-hand menu.



2. Select one of the three icons for workload types. A list of corresponding workloads will appear below.

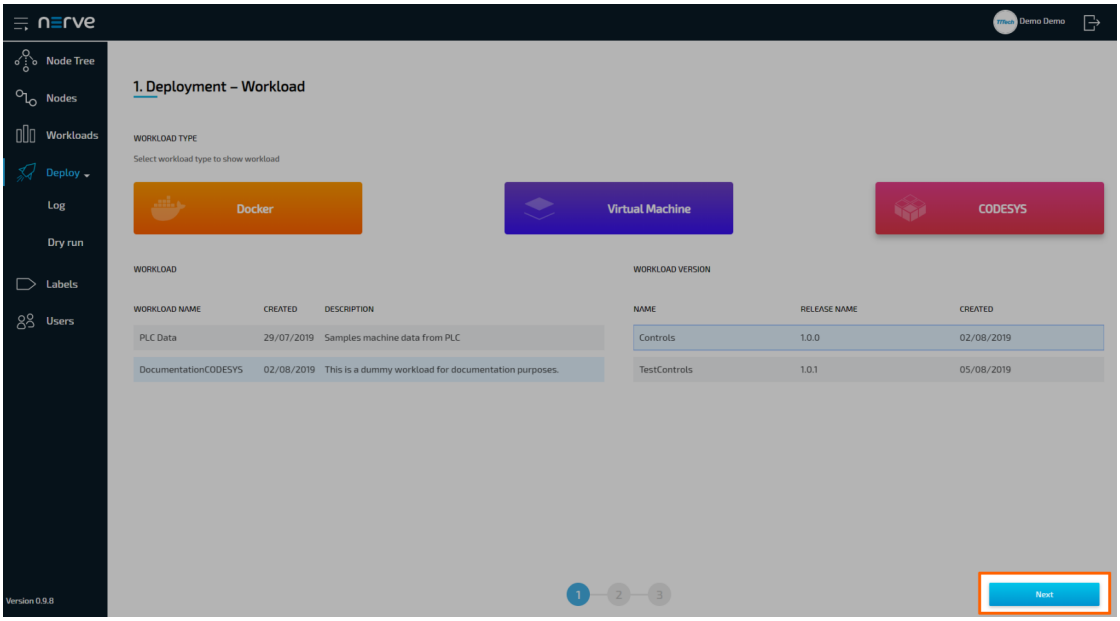


3. Select a workload from the list. A list of versions of this workload will appear to the right.



4. Select the version of the workload you would like to deploy.

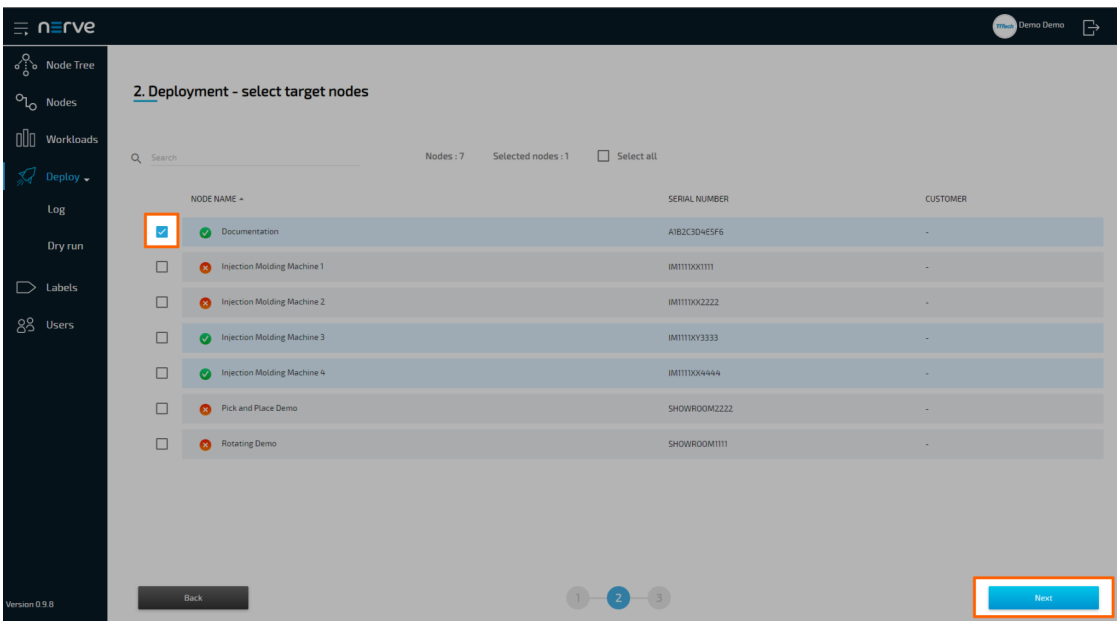
5. Click **Next** in the bottom-right corner.



6. In the next window, select one or more nodes from the list for deployment by ticking the checkboxes on the left.

> **NOTE**
>
> This list of nodes is automatically filtered depending on the labels the workload has. So this list of nodes might not include all nodes that are registered in the Management System.

7. Select **Next** in the lower-right corner.

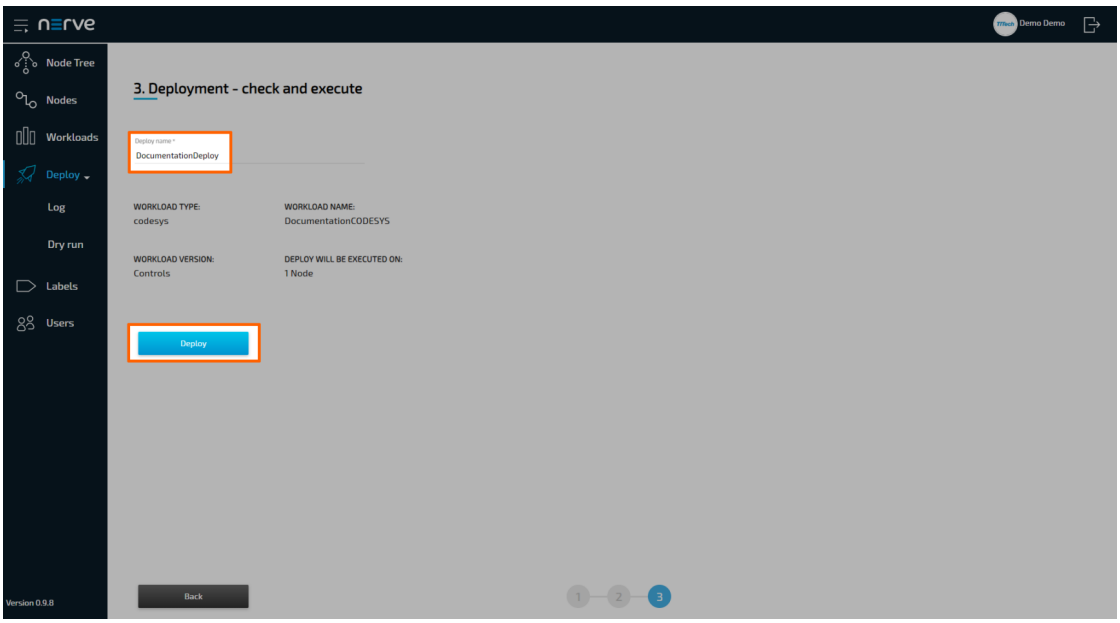


8. In the next window, enter a **Deploy name** above the **Summary** of the workload. Choose a name that makes this deployment easy to identify.

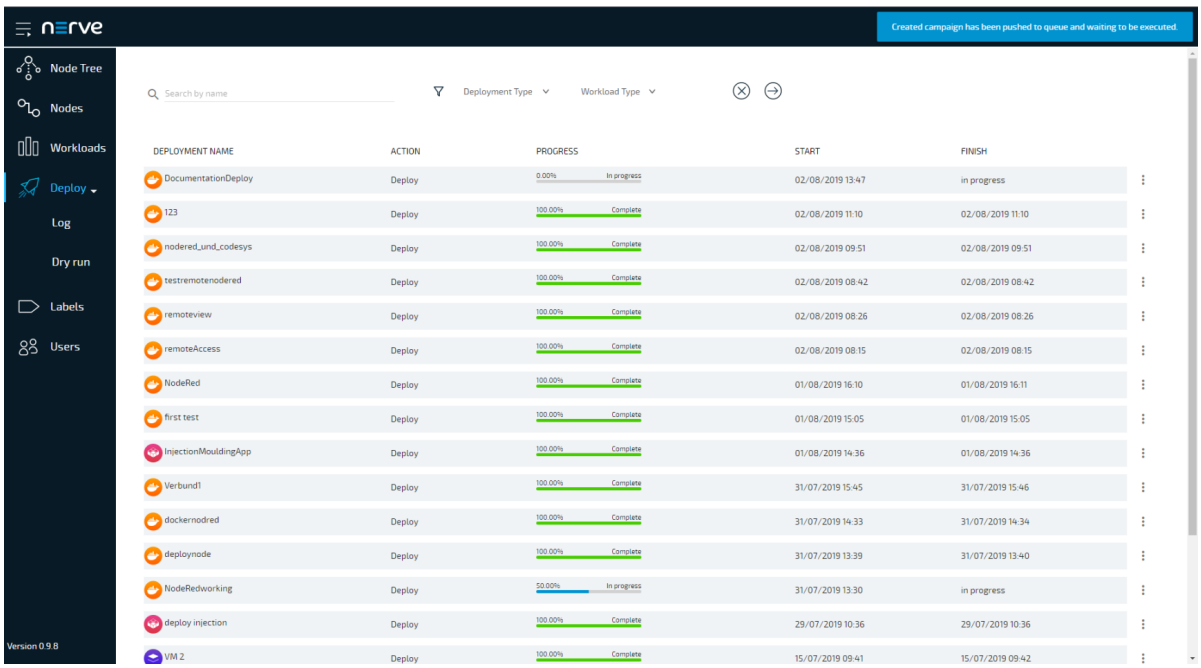
NOTE

You can only use alphanumeric characters (a-z, A-Z, 0-9) and underscore (_). Any other special characters are not allowed. If you use a character that is not allowed, the deploy button will be grayed out.

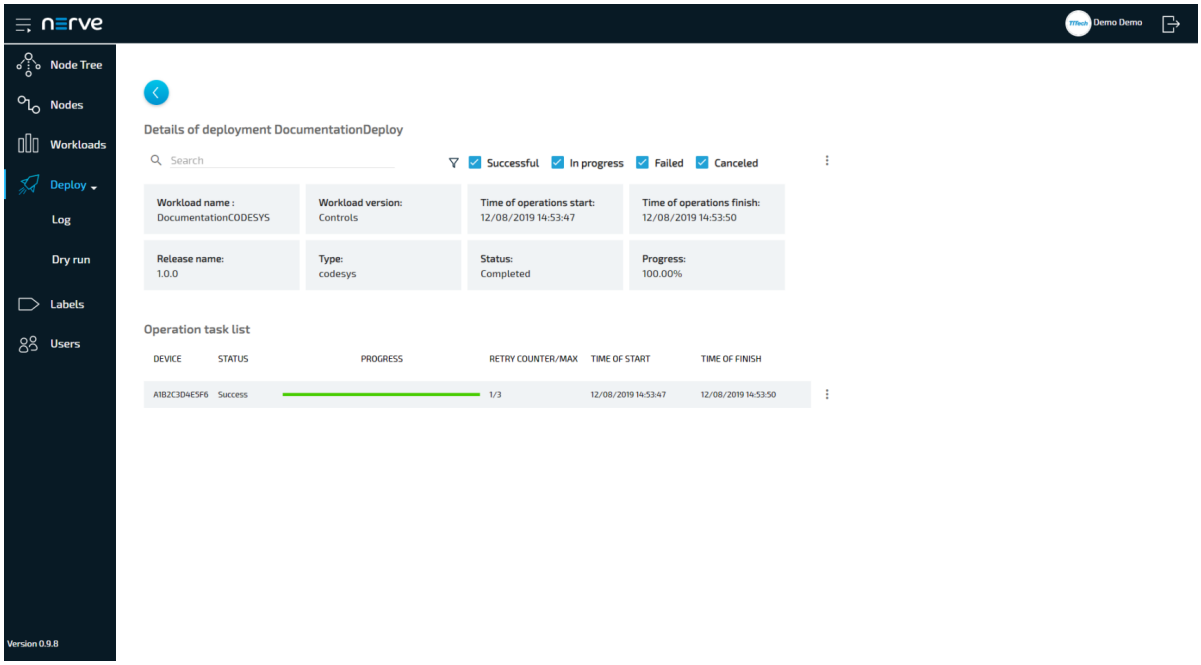
9. Select **Deploy** to execute the deployment.



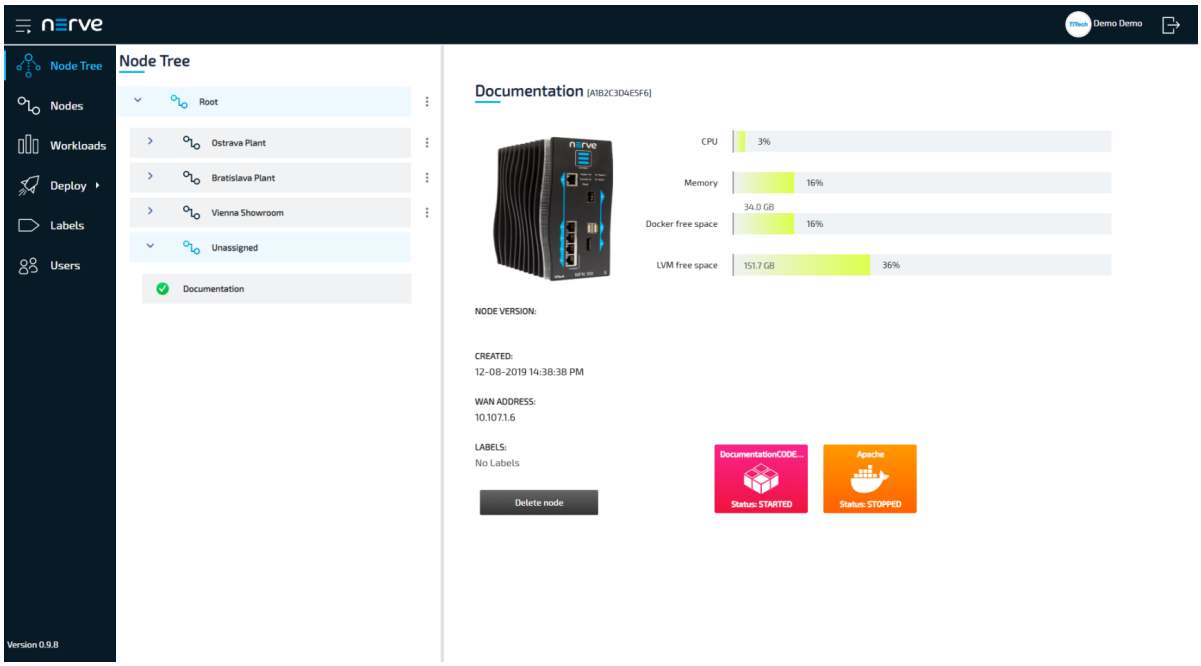
You will be taken to the log next. Your current deployment is at the top of the list. The**Deploy name** you have chosen before is the name that identifies the deployment in the log.



You can see the progress of the current deployment and click the log entry of the deployment to see a more detailed view.



The workload has been deployed and can be controlled in the Node Tree. Select **Node Tree** in the menu on the left and select the node you have deployed a workload to.



You can control the workloads by clicking the tile of each workload. All workloads are started as soon as they are deployed.

NOTE

CODESYS applications can only be controlled through the local UI.

Workload Example: Installing Node-RED

As a hands-on example, this chapter will guide you through provisioning and deploying Node-RED in a Docker workload with recommended settings. This is an example and a combination of the previous chapters that covers provisioning a Docker workload containing Node-RED and then deploying the provisioned Node-RED workload.

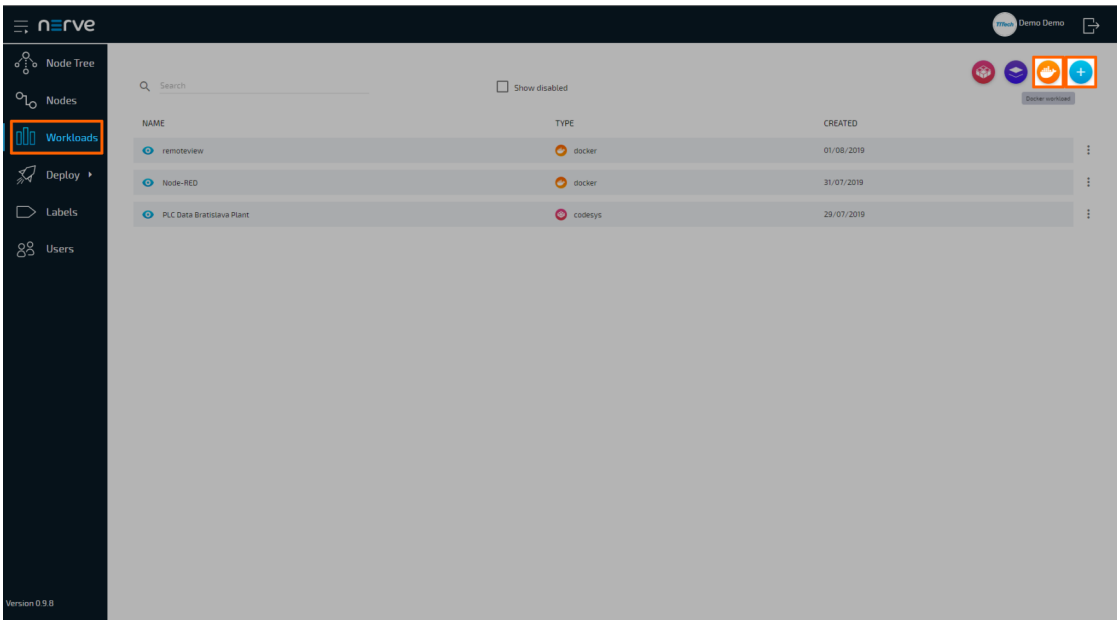
If you do not need Node-RED or if you feel confident in provisioning and deploying workloads, please feel free to skip this chapter.

Provisioning the Node-RED Workload

Before you can deploy a Node-RED workload, it needs to be provisioned first. Since this chapter provides a specific example, it will include recommended settings for the deployment of a Docker workload containing Node-RED.

If you want a more detailed look into the general process of provisioning a Docker workload, please refer to the chapter Working with Docker Containers.

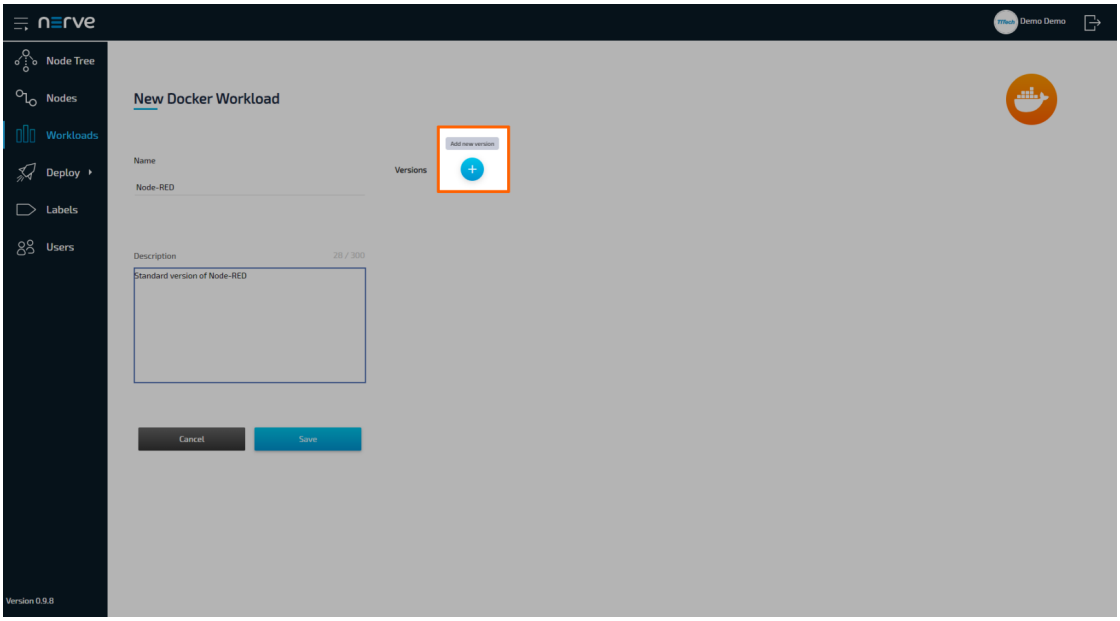
1. Log in to the Management System.
2. Select **Workloads** in the left-hand menu.
3. Select **+** in the upper-right corner.
4. Select the Docker symbol (**Docker workload**) on the right of the three symbols that expanded.



5. In the new window, enter a name for your workload.

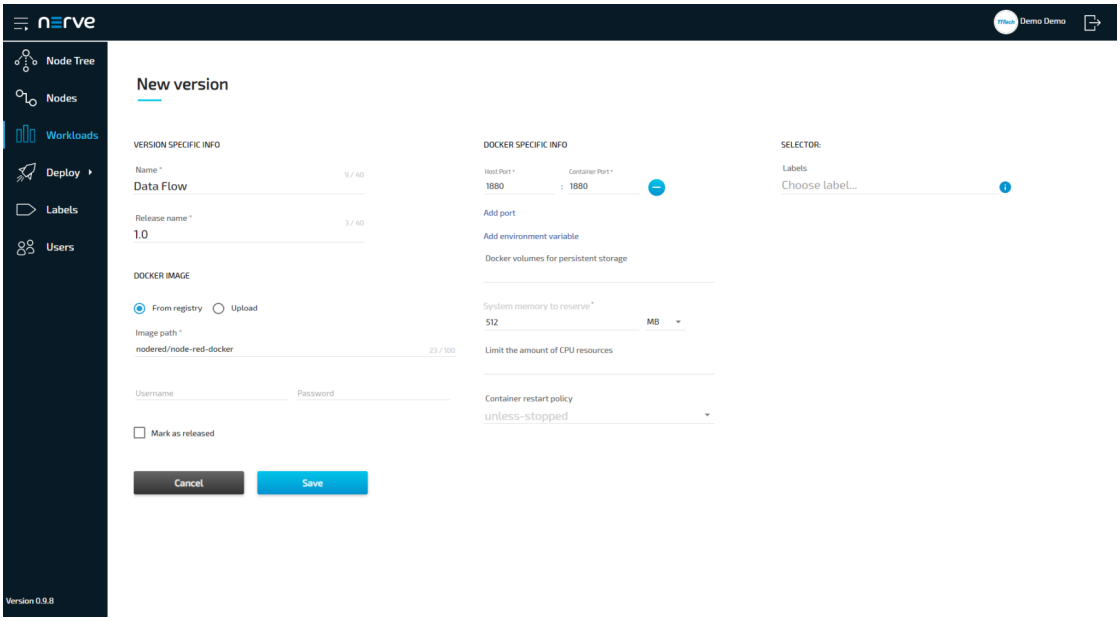
   Optional: Enter a description for your workload.

6. Select **+** next to **Versions** to add a new version of the workload.



7. In the next window, enter the following information:

Item	Description
Name	Enter a **Name** for the version of this workload. Here **Data Flow** is chosen to signify the functionality of this workload version.
Release name	Enter a **Release name** for the version of this workload. This can be a version number. **1.0** is used here as an example.
DOCKER IMAGE	Select **From registry** or **Upload**. If you select **From registry**, you need to enter the Docker registry to the Docker container image under **Image path**. For a Node-RED workload this is nodered/node-red-docker. If you select **Upload**, click the **upward arrow** symbol to upload the Node-RED Docker container image. Please refer to the Docker documentation if you need help with creating a Docker image.
Add port	Click here to define a **Host port** and a **Container port**. This setting is depending on the Docker container you are going to deploy. Therefore this is set to **1880:1880** as this is the default port that Node-RED listens on.
System memory to reserve	Assign how much system memory you would like to reserve for this workload. Here this is set to **512 MB**.
Container restart policy	Choose the container restart policy here to determine when the Docker container can be restarted. This is set to **unless-stopped** here so that Node-RED container only restarts if it is manually stopped.



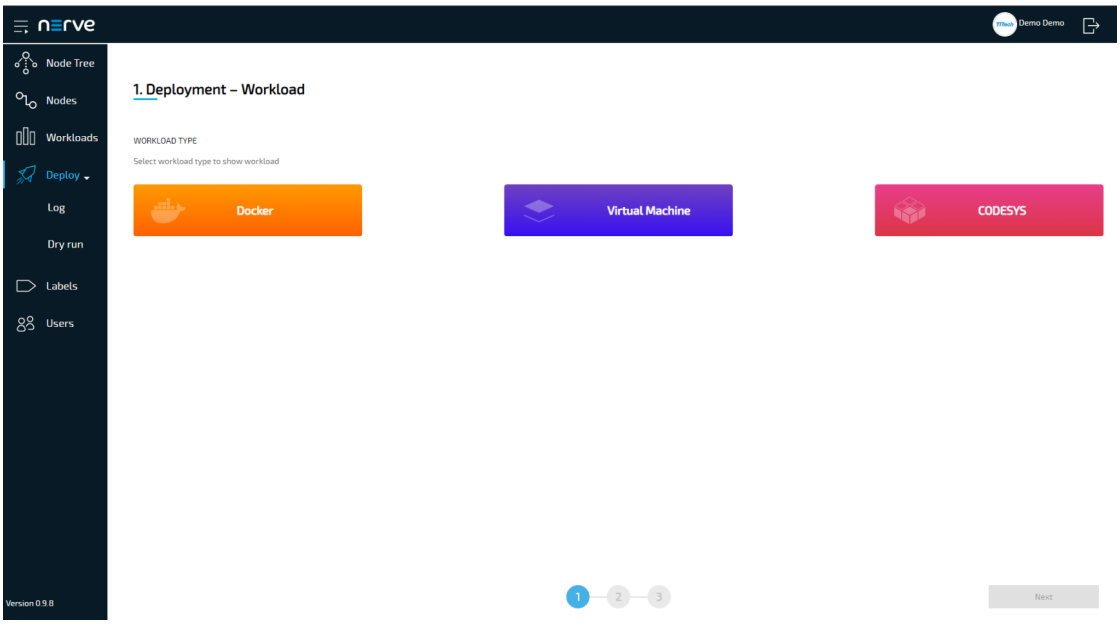
8. Select **Save** in the lower-left corner.

The Node-RED workload has now been provisioned and is ready to be deployed in the **Deploy** menu.

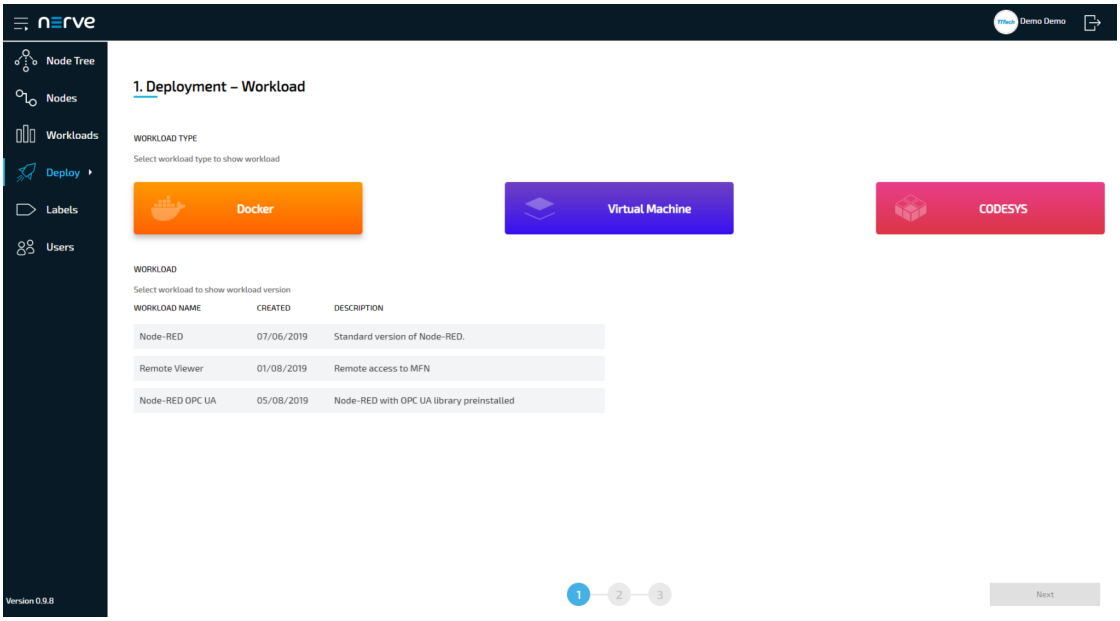
Deploying the Node-RED Workload

Now that the Node-RED workload is provisioned, you can deploy the workload to any designated node.

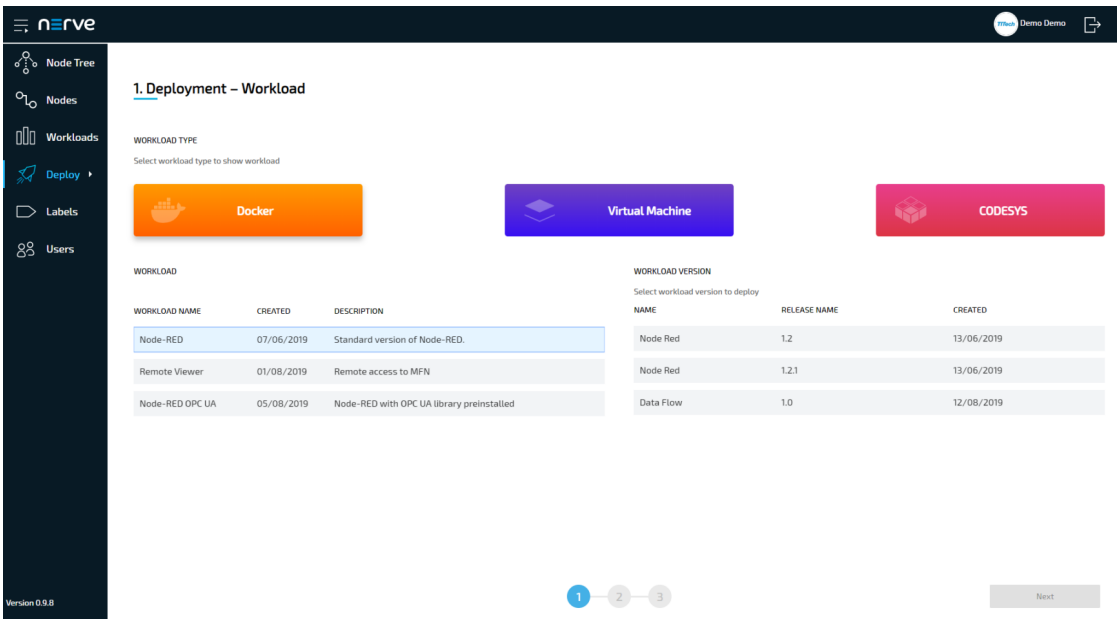
1. Select **Deploy** in the left-hand menu.



2. Select the Docker icon. A list of Docker workloads will appear below.

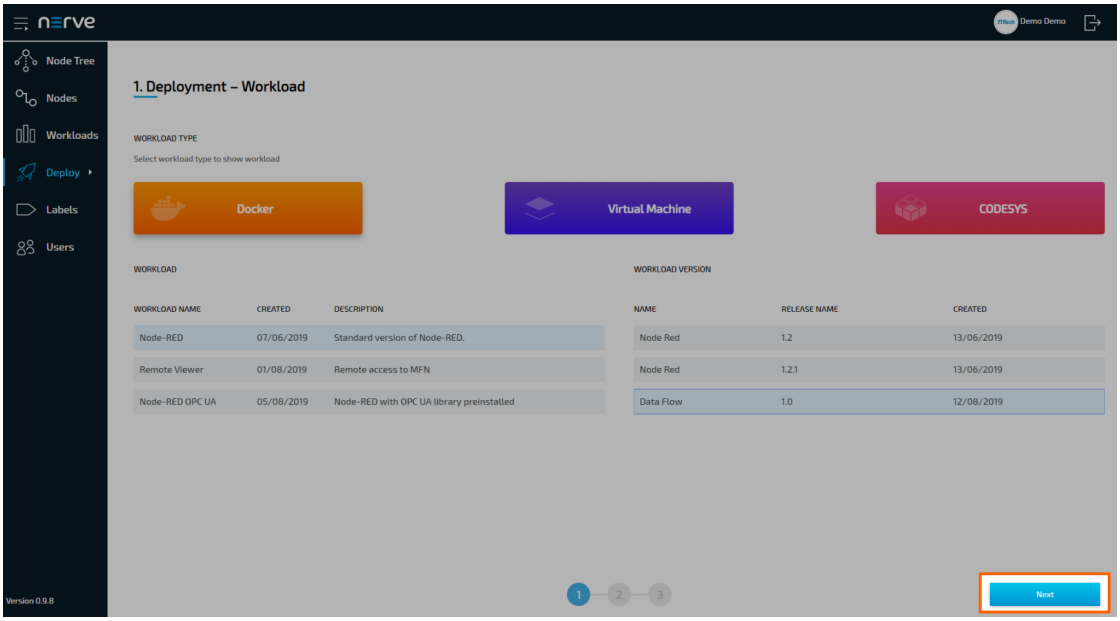


3. Select the **Node-RED** workload that you have provisioned before from the list. A list of versions of this workload will appear to the right.



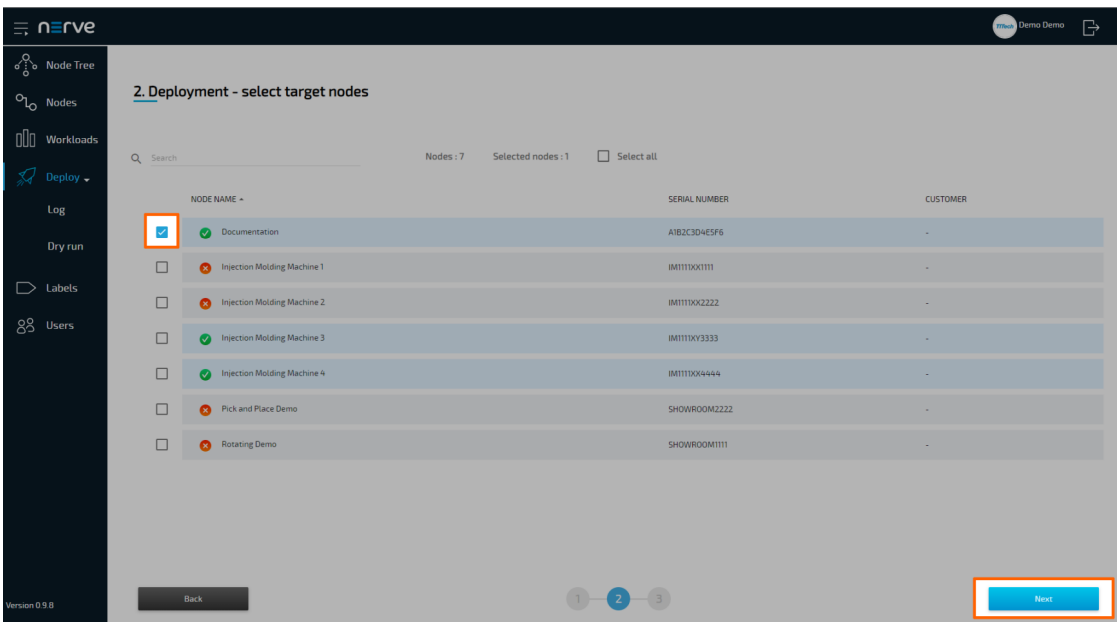
4. Select the version of the workload you would like to deploy. If you have followed this example, there will only be one version of this workload available.

5. Click **Next** in the bottom-right corner.



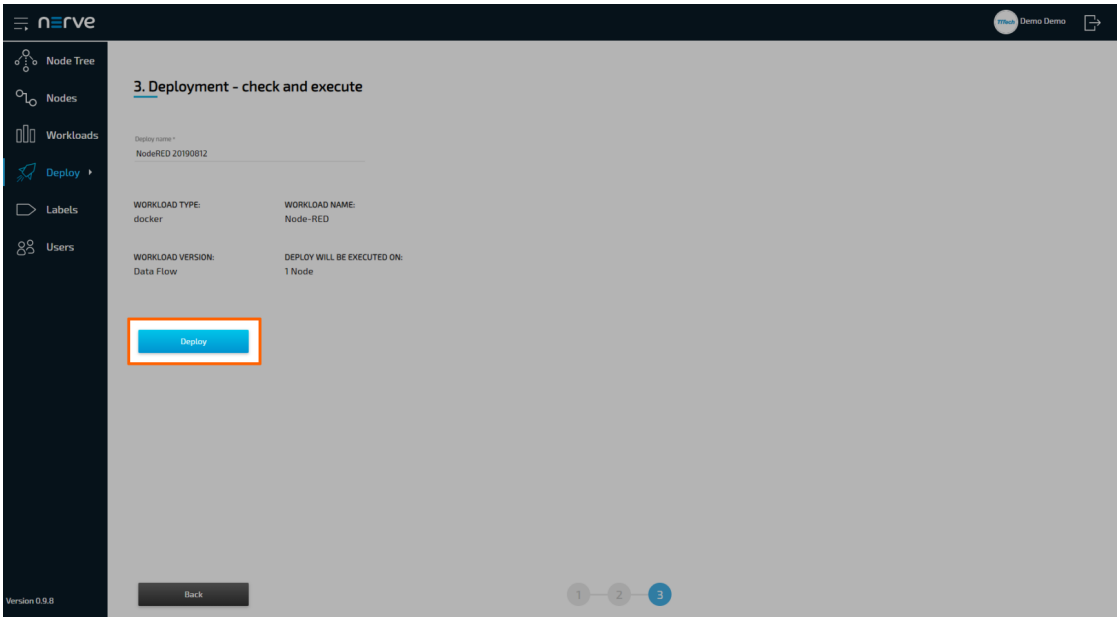
6. In the next window, select one or more nodes from the list for deployment by ticking the checkboxes on the left.

7. Select **Next** in the bottom-right corner.

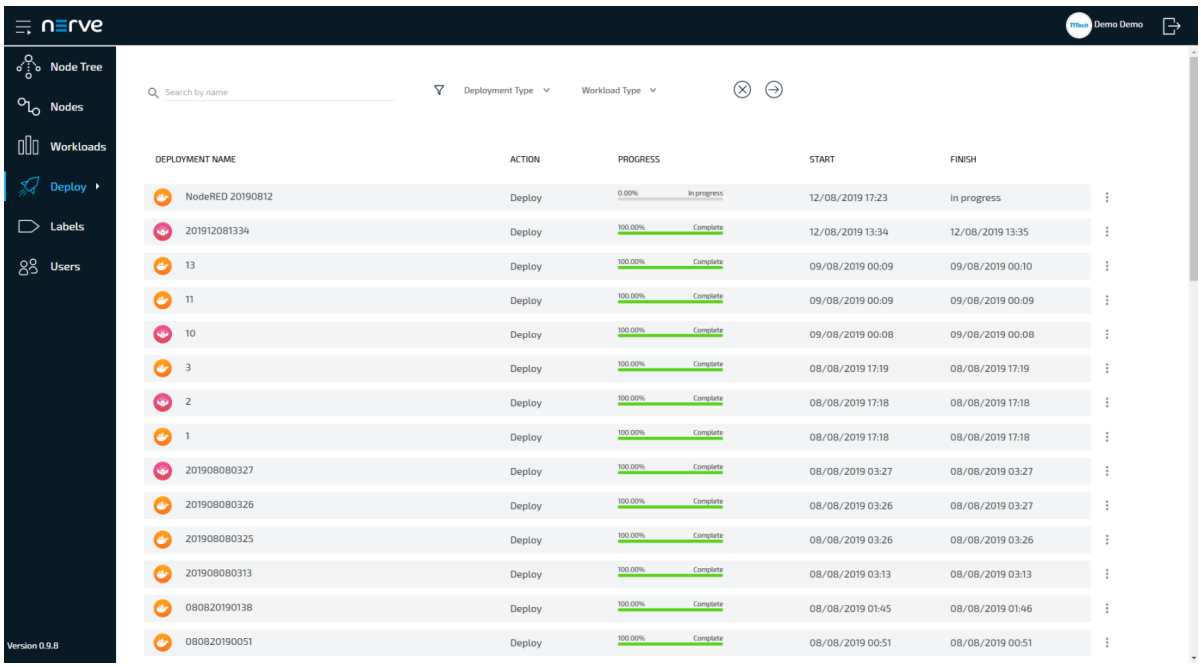


8. In the next window, enter a **Deploy name** above the **Summary** of the workload. Choose a name that makes this deployment easy to identify.

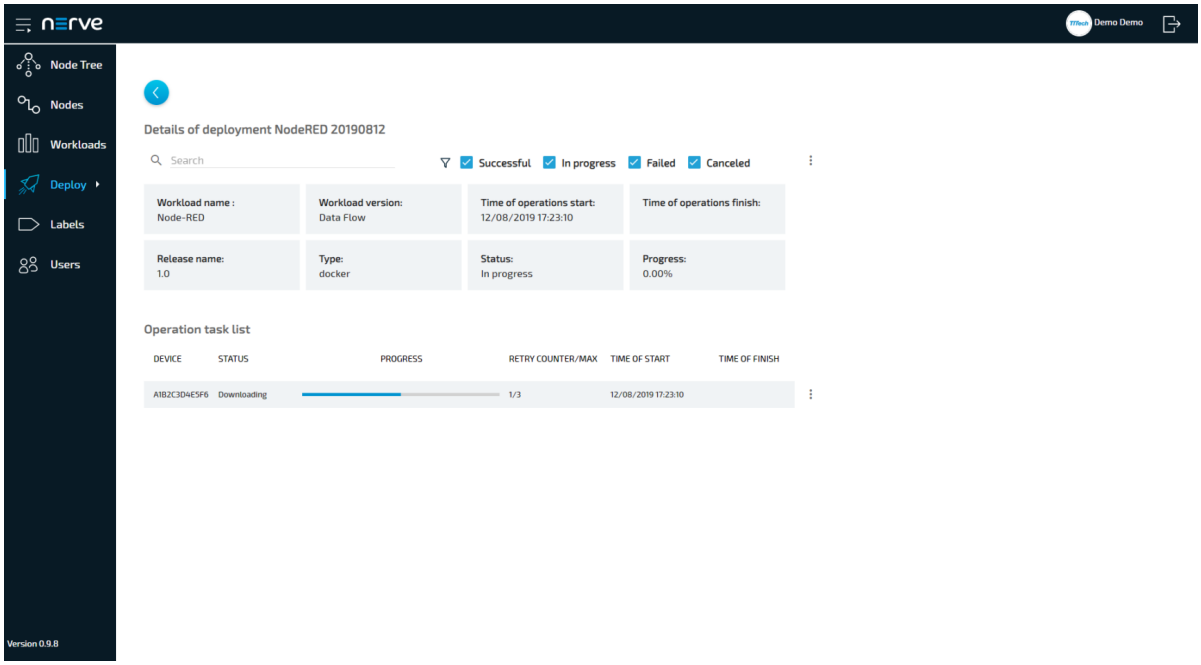
9. Select **Deploy** to execute the deployment.



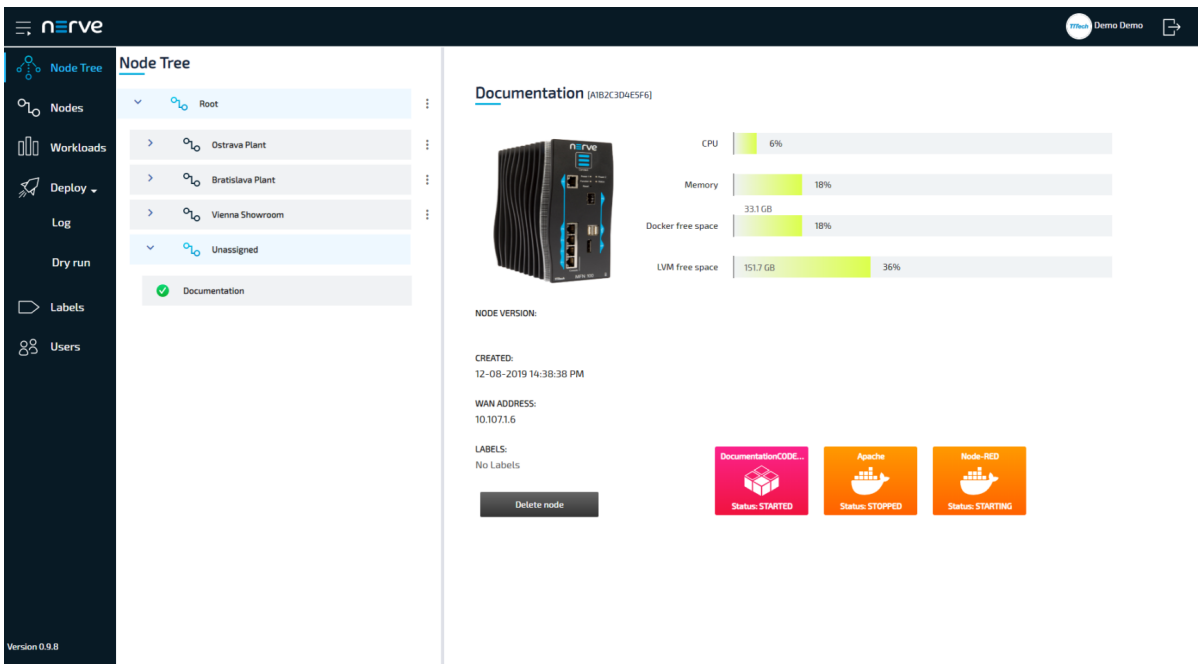
You will be taken to the log next. Your current deployment is at the top of the list. The**Deploy name** you have chosen before is the name that identifies the deployment in the log.



Here you can see the progress of the current deployment and click the progress bar to see a more detailed view.



The workload has been deployed and can be controlled in the Node Tree. Select a node to view the node details and choose the workload you would like to control. The Node-RED workload is started right after deployment.



This is the end of the quick start guide. If you have experienced any problems while following this guide or if you would like to give feedback, please contact your sales representative or TTTech Industrial customer support at support@tttech-industrial.com.

User Guide

User Guide

NOTE

If you want to get started quickly with Nerve Blue, please read through the quick start guide first.

The user guide for Nerve Blue covers the features and configurations a user has with Nerve Blue. This includes options in:

- the local UI
- and the Management System

This guide is followed by a developer guide that expands on the possibilities of Nerve relevant for creating workloads.

If you should have any questions about the software installed and the features provided, please contact your sales representative or TTTech Industrial customer support at support@tttech-industrial.com.

NOTE

Please have the customer profile ready. You are going to need it for the following chapters. It has been sent as part of the delivery. If you have not received a customer profile, please contact your sales representative or TTTech Industrial customer support at support@tttech-industrial.com.

Hardware Specifics

The user guide focuses on how to operate the Nerve Blue software. As such it will not contain any device specific information. Whenever device specific information is required, the user guide will link to the device guide.

In this version, the only available Nerve Device is the MFN 100. Other Nerve Devices will be added to the device guide as soon as they have been certified for usage with Nerve Blue.

Local UI

The local UI is provided by a web server that is running on the Nerve Device locally. Compared to the Management System, the local UI covers features that only concern the node itself. It serves three main purposes:

- Network configuration
- Control of CODESYS applications
- Password management

NOTE

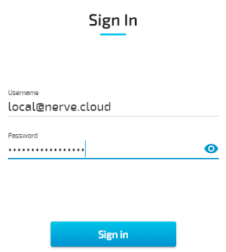
The credentials for the local UI are set in the node registration process. If the product has been configured by TTTech Industrial, you can find the credentials in the customer profile that has been sent as part of the delivery.

If you have not received a customer profile, please contact your sales representative or TTTech Industrial customer support at support@tttech-industrial.com.

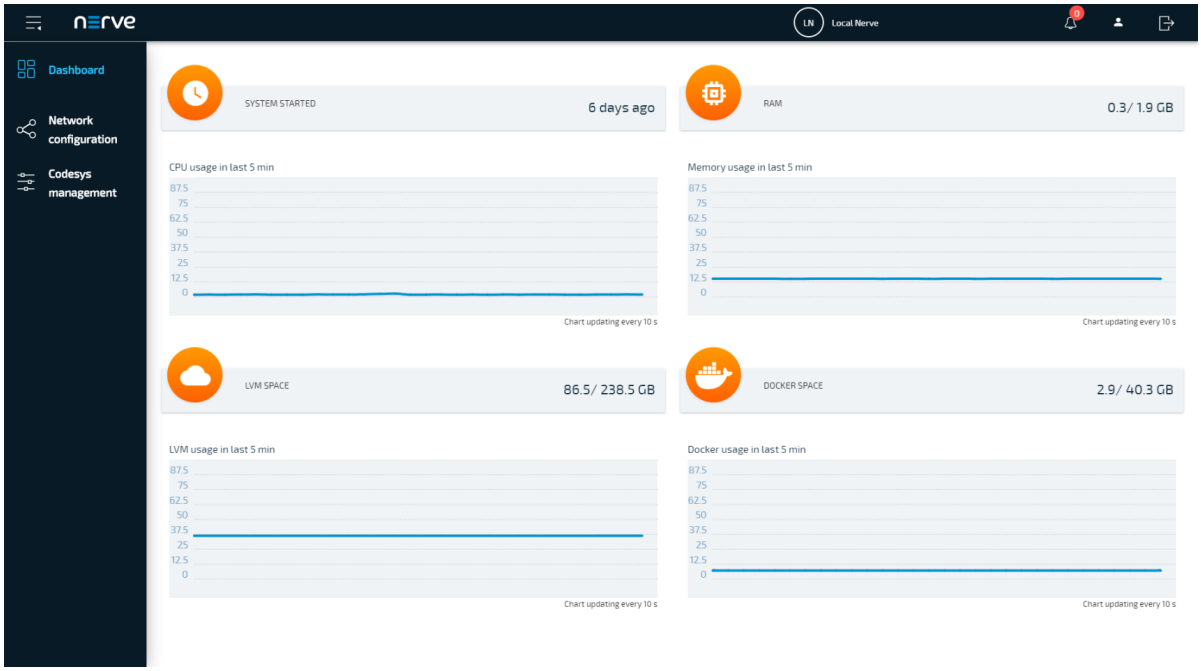
Connecting to the Local UI

Connecting to the local UI depends on your Nerve Device and your configuration. See the device guide for details on how to connect to the local UI of your Nerve Device.

Once you have established a connection, log in with the credentials for the local UI.

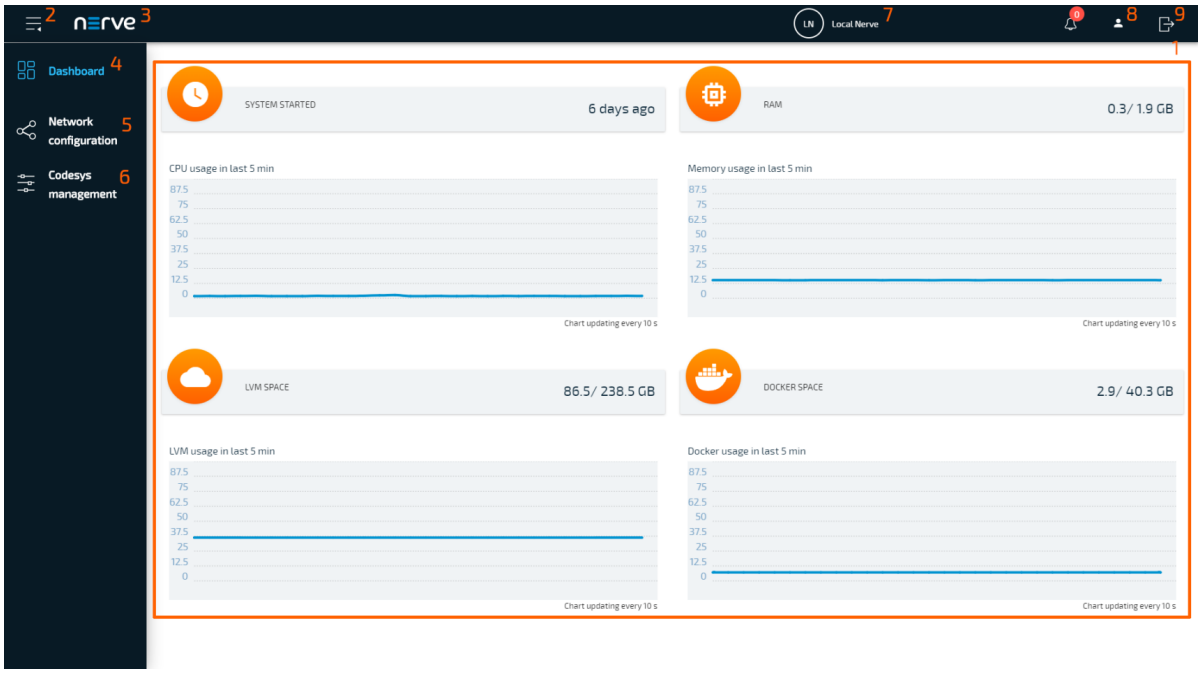


You will reach the main page of the local UI.



Local UI Dashboard

The dashboard of the local UI is the default screen after the log in. Here you can find usage statistics of your Nerve Device displayed and more options in the menu on the left side.



Term	Description
System metrics (1)	Graphs showing available resources of the Nerve Device and their usage over time.
Burger menu (2)	You can expand and collapse the left-hand menu by clicking here.
Nerve logo (3)	Click here to return to the dashboard and reload the page.

Term	Description
Dashboard (4)	Select this to display the dashboard.
Network configuration (5)	This menu allows you to configure the Ethernet ports of the Nerve Device.
CODESYS management (6)	Control options for deployed CODESYS workloads are here.
User name (7)	This is the username of the signed in user.
Change password (8)	Clicking here leads to an area where you can change the password to the local UI.
Log out button (9)	Click here to log out of the local UI.

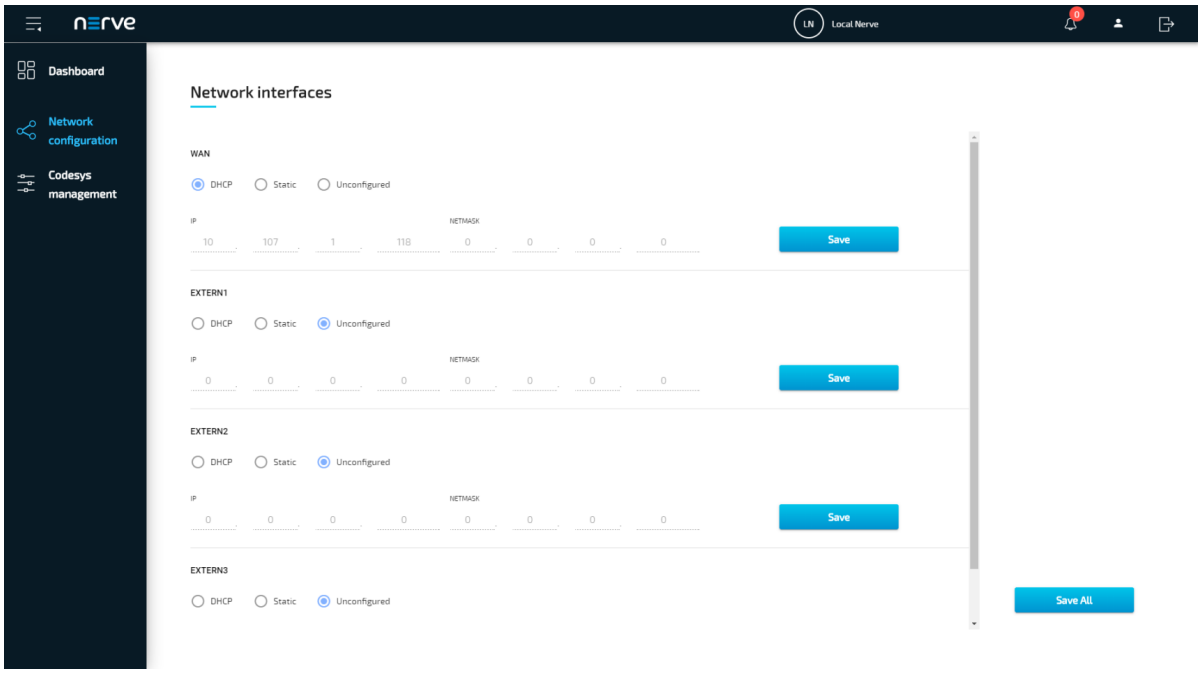
System Metrics

The graphs in the local UI dashboard show available resources of the Nerve Device and their usage over time. The y-axis displays percentages and the x-axis is updated every 10 seconds, showing a time span of 5 minutes. The percentages displayed are always in relation to the maximum of the available resource:

Item	Description
SYSTEM STARTED	This shows how long the Nerve Device has been running. If the device is restarted, this value is reset.
CPU usage in last 5 min	The graph here shows the percentage of processing power that is being used. This includes CPUs that have been assigned to VMs and Docker Containers.
RAM	This shows how much memory is used (left value) and how much memory is available in total (right value). Example: 0.3/1.9 GB Please note that the total amount of memory your Nerve Device offers is not shown here. This is the memory that is available for the host.
Memory usage in last 5 min	Similar to CPU usage, the graph shows the percentage of memory used. This includes memory that has been assigned to VMs or Docker Containers.
LVM SPACE	Virtual machines have their dedicated virtual partition (Logical Volume Manager). The values show how much of this partition is used (left value) and how much is available in total (right value). Example: 86.5/238.5 GB
LVM usage in the last 5 min	This graph shows the percentage of space that is being used by the Logical Volume Manager.
DOCKER SPACE	Similar to LVM space, Docker containers have their dedicated virtual partition. The values show how much of this partition is used (left value) and how much is available in total (right value). Example: 2.9/40.3 GB
Docker usage in the last 5 min	This graph shows the percentage of space for Docker containers that is being used.

Local Network Configuration

From the local UI, you can configure the Ethernet ports of the Nerve Device. Select **Network configuration** in the navigation on the left to reach this menu. The example below is of the MFN 100. The page is specific to your Nerve Device. The number and names of interfaces may differ.



The options below are available for every interface:

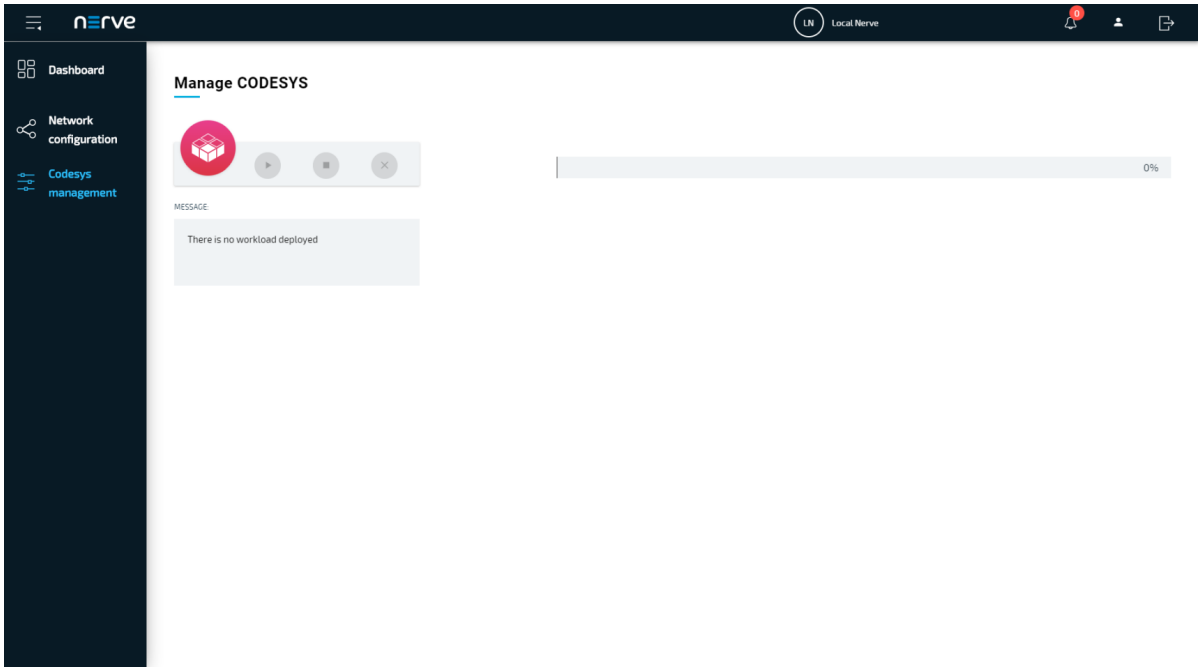
Item	Description
DHCP	The IP address of the port will be assigned by the DHCP server. If an IP address has been assigned, it will be displayed here.
Static	By selecting **Static**, you have to define the IP address of the port manually. Enter the IP address here to set a static IP address.
Unconfigured	If **Unconfigured** is checked, the port is disabled for the host but can still be used for virtual machines with bridged interfaces.

NOTE

For more information on networking and interfaces please refer to the networking chapter.

Control of CODESYS Applications

CODESYS workloads can only be controlled in the local UI, as operation of a CODESYS workload may have an impact on your machine operation and therefore should not be controlled remotely. Select **CODESYS management** in the menu on the left-hand side to reach the interface for controlling a CODESYS application running on the Nerve Device:



Function Name	Description
Start	This starts the CODESYS application.
Stop	This stops the CODESYS application and it is reset to its initial values.
Remove	This removes the CODESYS application from the Nerve Device. If you want to deploy the CODESYS application again, you have to do so directly with local access through the CODESYS Development System or through the Management System in the cloud.
Message	CODESYS workloads have the following set of messages: • "Preparing files for installation" • "Starting CODESYS application" • "CODESYS application started" • "Stopping CODESYS application" • "CODESYS application stoppped" • "Removing CODESYS application file" • "An unexpected error has occurred. <errormessage>" Here, <errormessage> is a message that is sent by the CODESYS Development System.

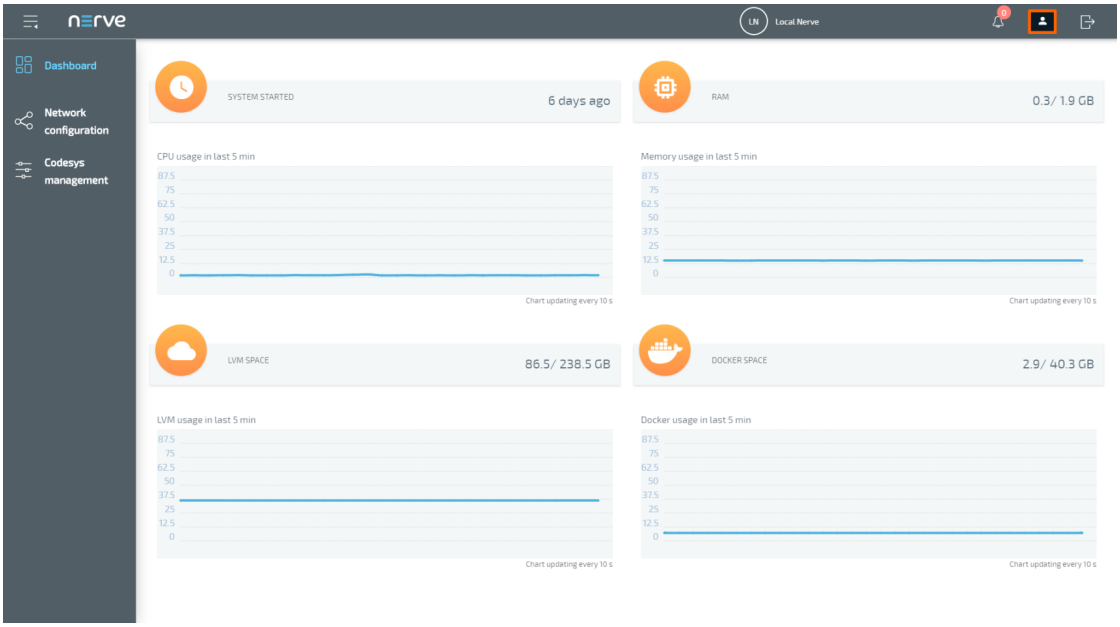
NOTE

It takes a moment before CODESYS applications are actually started, stopped or removed.

Password Management

You can change the password to the local UI if you wish.

1. Select the user icon (**Change password**) in the upper-right.

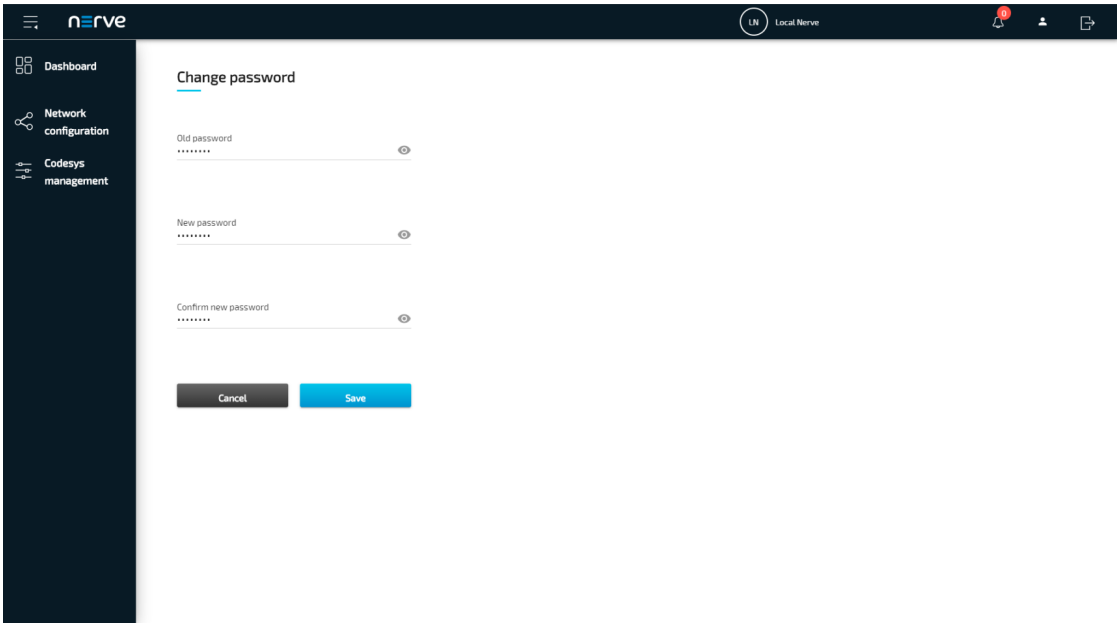


2. Enter the following information:

Item	Description
Old password	Enter the old password to the local UI.
New password	Enter the new password here. The new password must be 8 characters or longer and it can only consist of alphanumeric characters.
Confirm new password	Enter the new password again. Both passwords must match in order to proceed.

NOTE

The password to the local UI can also be set in the node registration process. See the node registration chapter for more information.

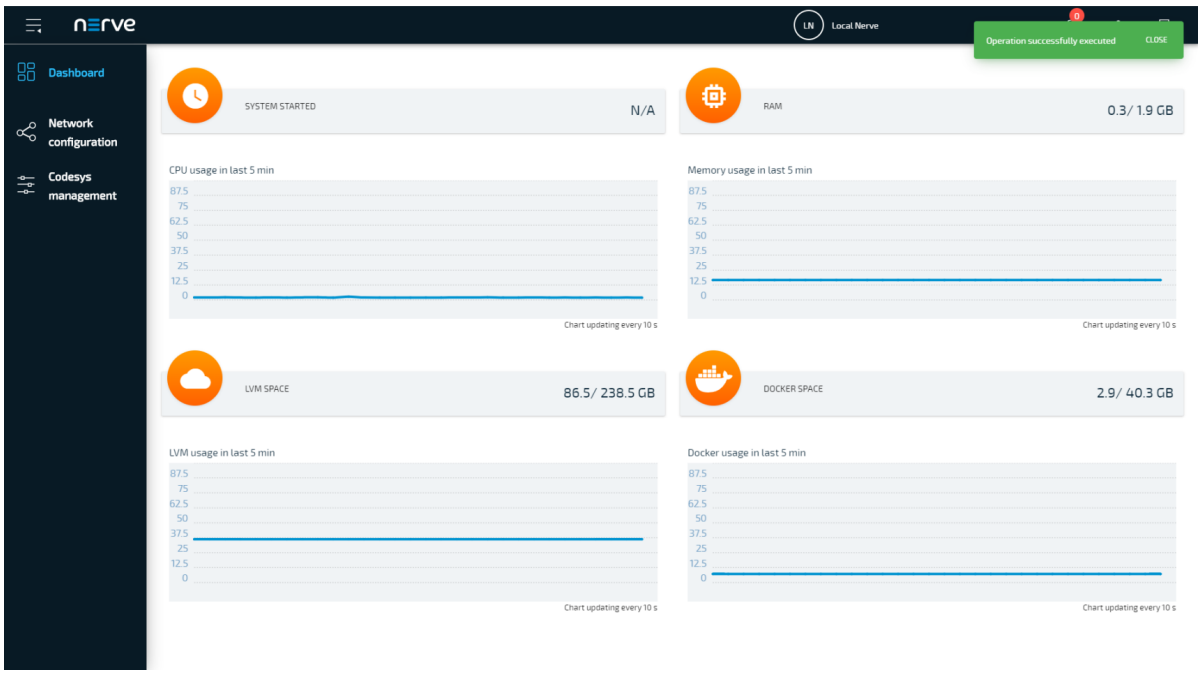


NOTE

The save button will stay grayed out until the fields are filled in correctly.

3. Select **Save** to set your new password.

If the process was successful, you will be returned to the dashboard with a green notice confirming the change in the upper-right corner.



Nerve Management System

Nerve Management System

The Nerve Management System is a web-based service that permits management of Nerve Blue nodes that are registered. It can be used to:

- Monitor nodes
- Deploy and control workloads on a node
- Manage workloads

NOTE

Google Chrome or Firefox Version 63 or later are recommended for the usage of the Management System.

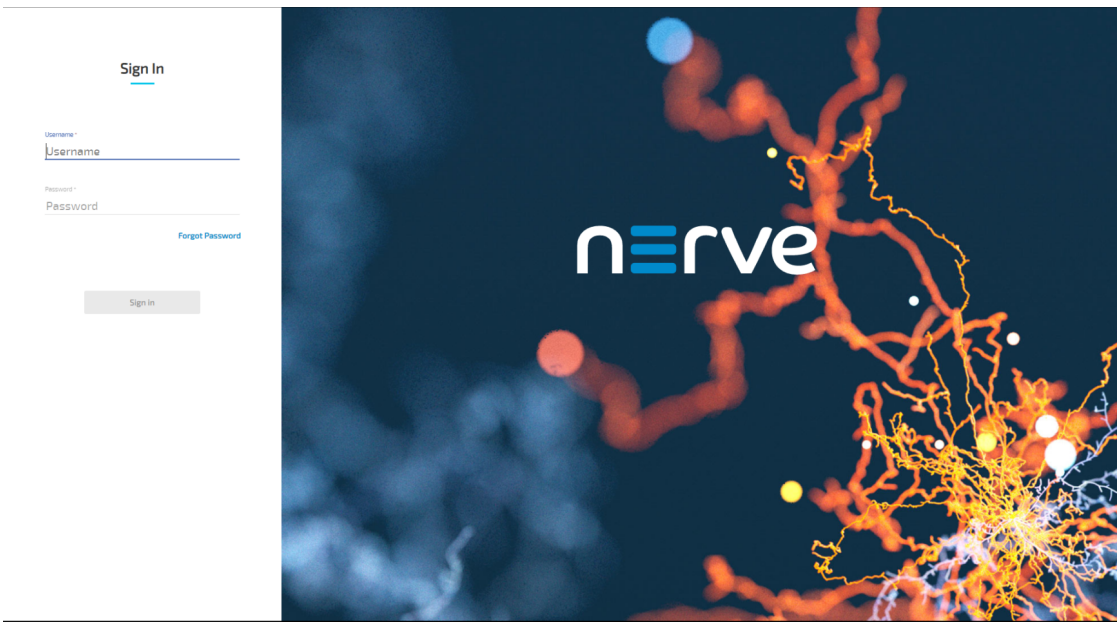
Logging in to the Management System

The Management System is either hosted by TTTech Industrial or installed on your premises. The URL of the Management System changes accordingly. You can find the URL to your Management System in the customer profile if it is hosted by TTTech Industrial.

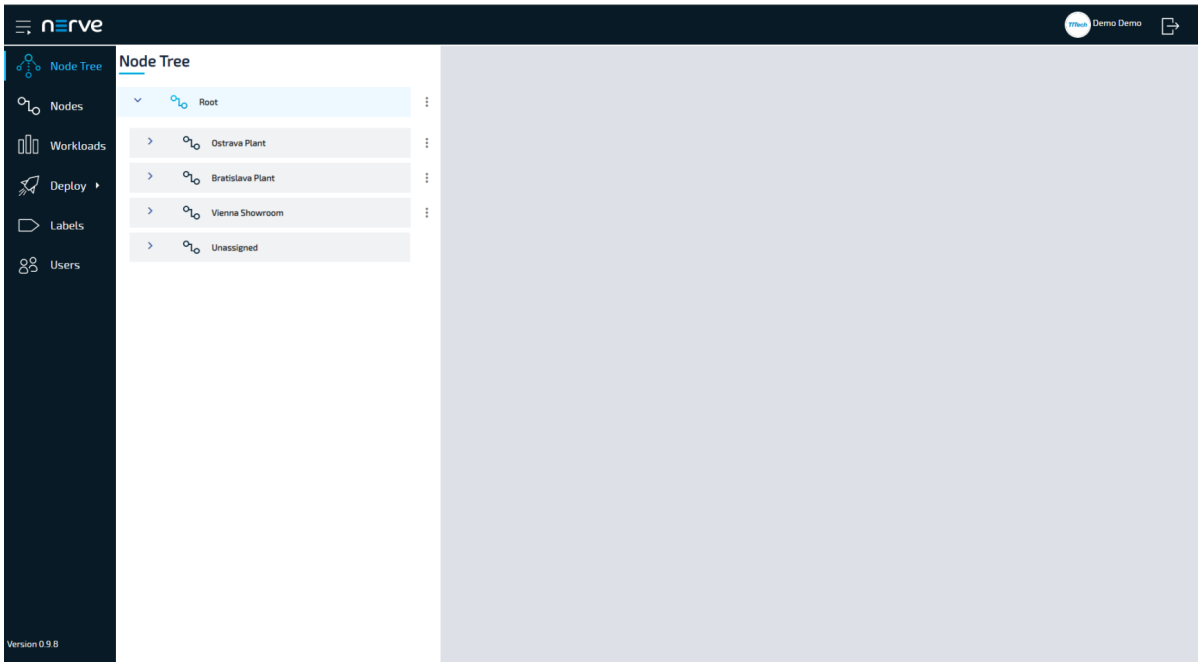
1. Go to the URL of the Management System.
2. Log in with the credentials for the Management System.

NOTE

The initial login credentials of the Management System can be found in the customer profile. If you have not received a customer profile, please contact your sales representative or TTTech Industrial customer support at support@tttech-industrial.com.

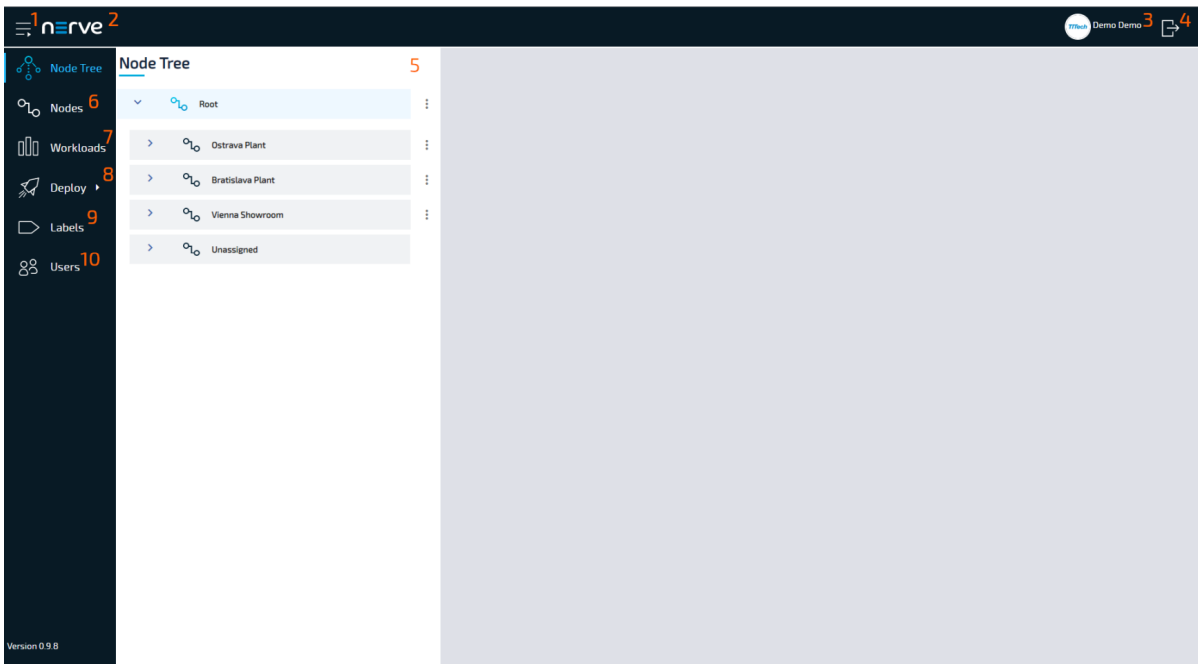


You will reach the Node Tree in the Management System.



Menu Structure of the Management System

As a landing page the Management System shows the Node Tree. You can navigate the site by using the menu bar on the left side.

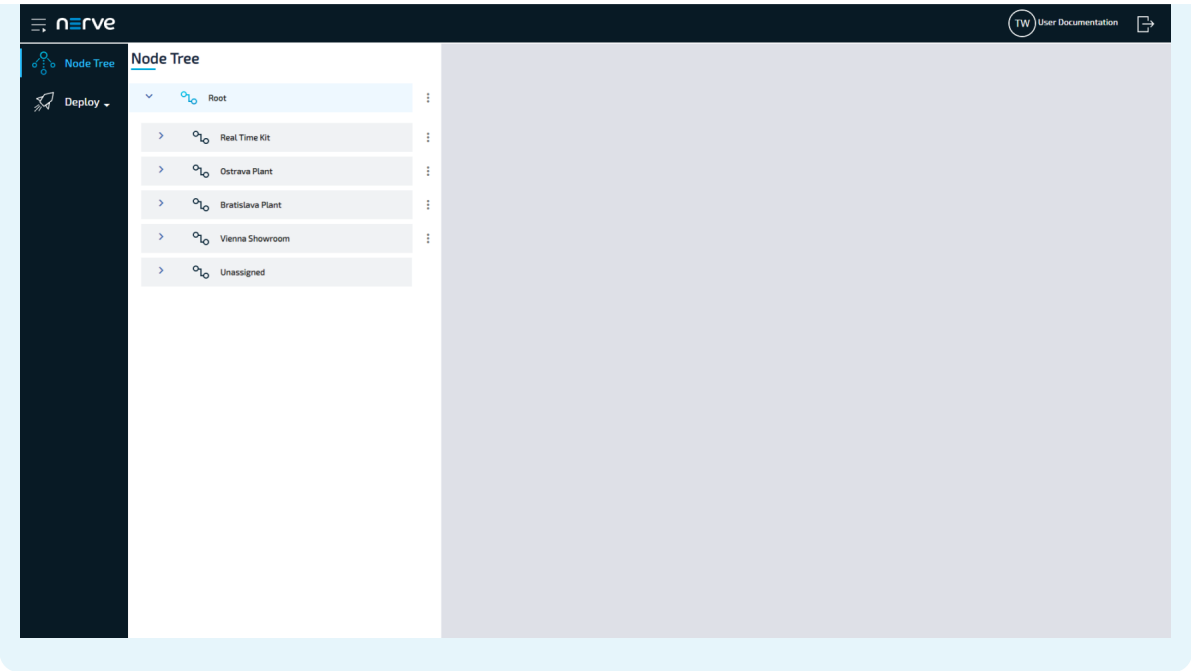


Item	Description
Burger Menu (1)	Clicking here will expand and collapse the menu bar on the left side. The expanded version of the menu adds names to the menu icons.
Nerve logo (2)	By clicking the Nerve logo, you can return to the Node Tree.
User display (3)	From here you can access your user profile and edit its details.
Log out button (4)	Clicking this icon will log you out of the Management System.

Item	Description
Node Tree (5)	This is the default view of the Management System. It displays all registered nodes in a Node Tree. You can add, delete and move tree elements freely here, as well as manage nodes. It mainly serves an organizational purpose and does not impact the functionality of the nodes.
Nodes (6)	Here you can see a list of all available nodes that have been registered in the Management System. You can add and remove nodes, as well as edit their details.
Workloads (7)	All workloads that have been provisioned in the Management System are listed here. Workloads can be added, deleted, disabled and edited through this menu. New versions of workloads can also be added here.
Deploy (8)	Workloads that have been provisioned previously can be deployed by following the menu on the right. However, there are two extra menus available here: • **Log** Here you will find a list of all workloads that have been deployed or are currently being deployed. You can also view details of all deployments and delete log entries from the list. • **Dry Run** A dry run is a simulation of a workload deployment. It allows you to test out if the deployment of a workload could be successful. However, please note that a successful dry run is not a guarantee for successful deployment.
Labels (9)	This is a list of all labels that have been defined in the Management System. In this menu you can add, delete, edit and merge labels.
Users (10)	This is the user management menu. It lists all registered users and allows you to edit profiles and add new users.

NOTE

Only users that are assigned an admin role can access all functionality. Therefore, the available features depend on your role. Regular users can only access the **Node Tree** and **Deploy** menus. They can also customize their own user profile.



Registering a Node in the Management System

In this version, it is required to register the node through a Python command line tool before you can use it in the Management System. The command line tool will be executed locally on the Nerve Device.

During the registration process there are two key parts happening: First, the node is configured. This means that the serial number is assigned and its secure ID is generated according to its serial number. The login credentials for local UI, host and RT VM access are also set. Second, the node is registered in the Management System with its serial number and secure ID so that the node can log in when communication is needed. For that, an existing admin user in the Management System is required. The admin user admin@nerve.cloud is defined by default.

NOTE

The files have been sent as part of the delivery. If you have not received the required files for the registration of a node, please contact your sales representative or TTTech Industrial customer support at support@tttech-industrial.com.

Hardware Setup

In order to connect the node to the Management System, you will need to set up the Nerve Device and connect your local workstation to the Nerve Device:

1. Please set up your Nerve Device as described in the device guide.
2. Connect your workstation to the physical port associated with host access.

NOTE

Please refer to the device guide for details on host access with your Nerve Device.

With this, the hardware is ready for the node registration process.

NOTE

Port 443 (HTTPS) and port 8883 (MQTTS) of the corporate firewall have to be open for communication between nodes and the Management System.

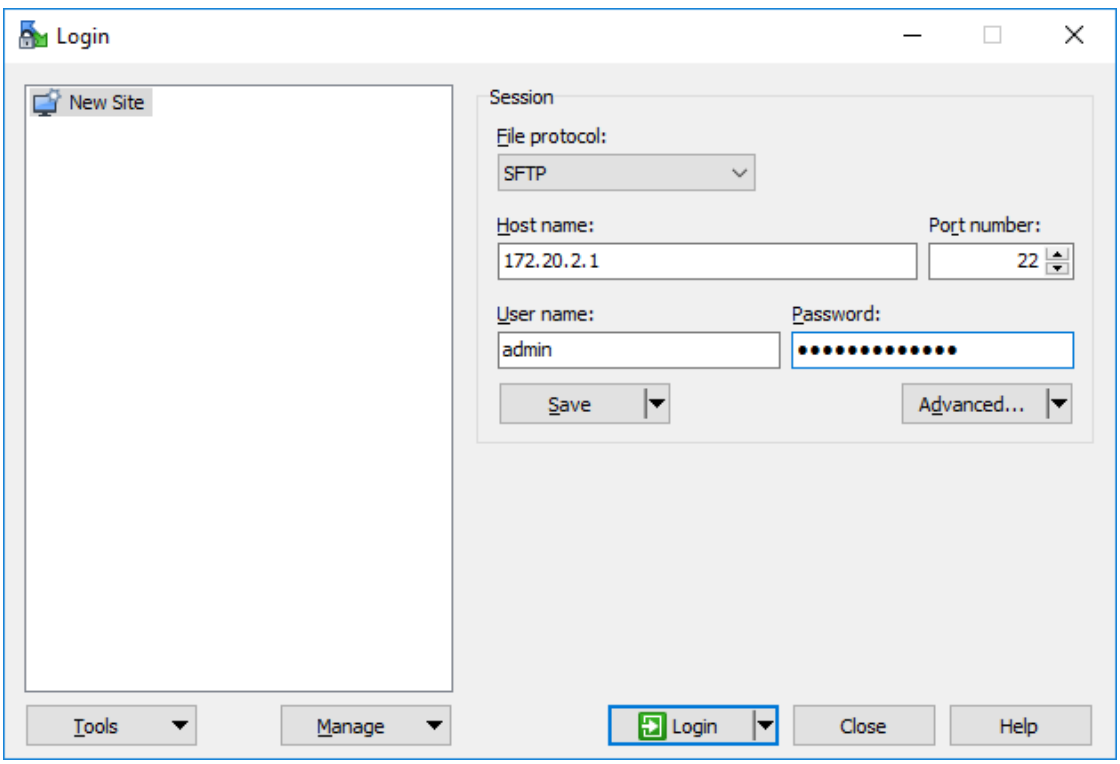
Copying the Tool to the Nerve Device

As the dependencies for the command line tool are already installed on the Nerve Devices, you only need to copy it to the Nerve Device and execute it from there. To do so, you will need an SSH client and a file transfer client. PuTTY and WinSCP are used as examples.

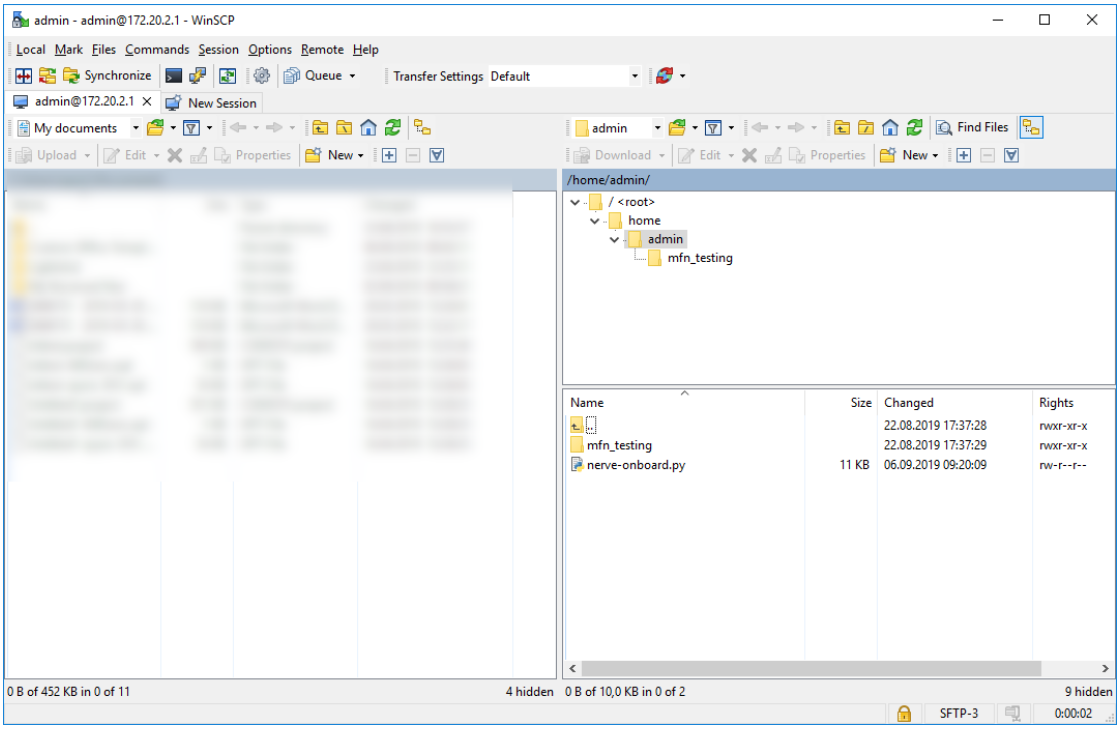
NOTE

The screenshots below use the MFN 100 as an example. Please refer to the device guide for the appropriate information on your Nerve Device.

1. Open a file transfer client like WinSCP.
2. Enter the host IP address of your Nerve Device under **Host Name**.
3. Enter the credentials for host access to the Nerve Device below under **User name** and **Password**.



4. Copy the nerve-onboard.py to the Nerve Device. The default path is /home/admin/ on the Nerve Device.



With the command line tool now copied to the Nerve Device, you can execute the command line tool locally on the Nerve Device. We will use an SSH connection to do so.

Using the Tool

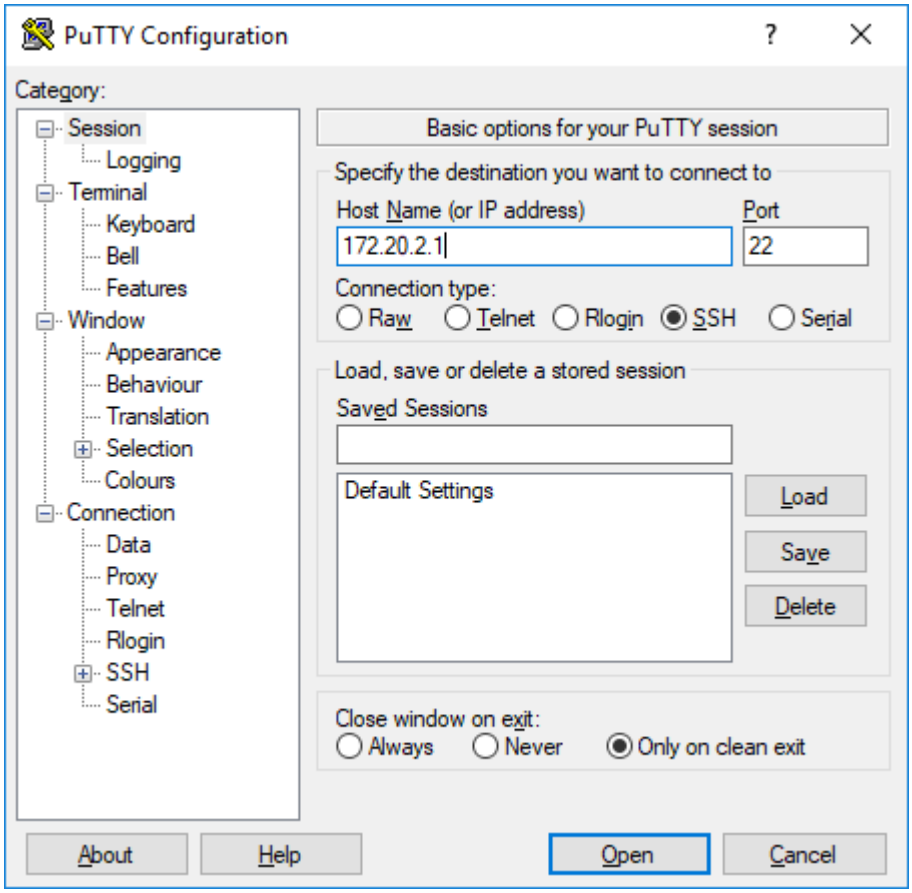
Now that all preparations are complete, you can execute the command line tool and start the registration of the node in the Management System.

NOTE

In the instructions below it is assumed that the default admin user is available in the Management System.
If you have already created new users and the default admin user is not available anymore, please add the cloud user parameter to the command and specify the user that will be used instead to log in to the Management System: python3 nerve-onboard.py --cloud-user <username> <domain> <serial>

1. Open an SSH client like PuTTY.

2. Enter the host IP address of your Nerve Device under **Host Name (or IP address)**.



3. Click **Open**.

4. Log in with the credentials for host access of the Nerve Device.

5. Enter python3 nerve-onboard.py <domain> <serial> and replace the placeholders with this information:

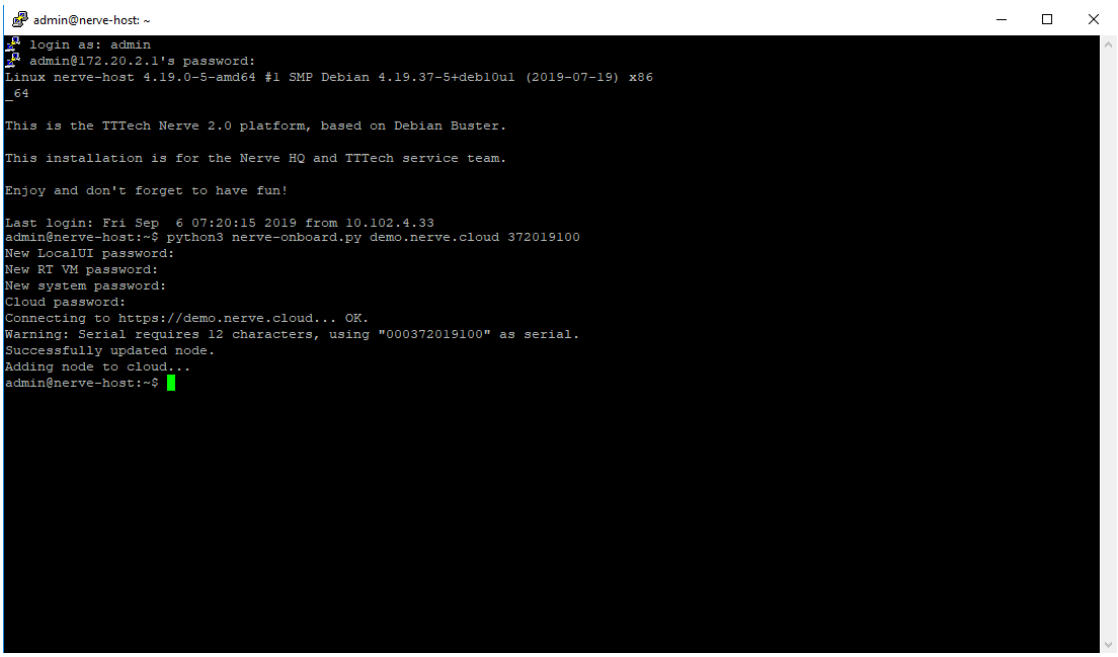
Item	Description
<domain>	This is the URL of your cloud. Please enter the URL without the protocol in the beginning. You can find it in the customer profile. Example: example.nerve.cloud
<serial>	Enter the serial number of your Nerve Device here. You can find it on the label of the Nerve Device.

NOTE

Depending on your Nerve Device, the command line tool might extend the serial number to 12 characters. Please make sure to take note of that after registering a Nerve Device.

6. Enter the following information in the table. The command line tool will prompt you one by one:

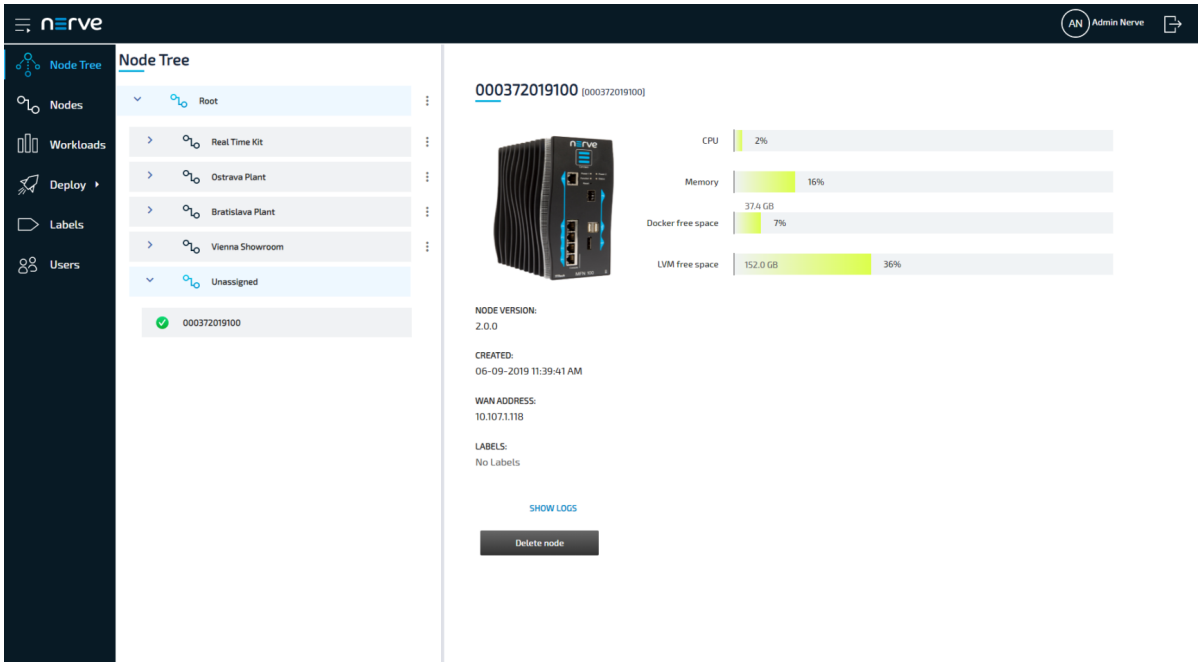
Item	Description
New LocalUI password	Define the new password for the local UI of your Nerve Device.
New RT VM password	Define the new password for the CODESYS runtime.
New system password	Define the new password for SSH host access.
Cloud password	Enter the password of your cloud access.



NOTE

The passwords above must be at least 8 characters long.

The node is now online and registered in the Management System. You can find it under **Root > Unassigned** in the Node Tree.



NOTE

Please make sure to note down the new passwords you have defined as you will need them later.

Using the Tool Locally

Since the dependencies for this tool are installed on the Nerve Device, the preferred method is to use the command line tool on the Nerve Device. However, the tool can still be executed locally if you prefer.

The tool needs Python 3 and Requests. Please download and install Python 3 on your workstation and follow the instructions on the Requests homepage to install Requests. On Debian and Ubuntu, enter apt-get install python3 python3-requests to install all dependencies.

Navigate to the folder where nerve-onboard.py is located on your workstation and open a command line. After that, the steps for using the tool are identical to the steps above. Follow the steps above starting from step 5 to register a node successfully.

Command Line Interface Parameters

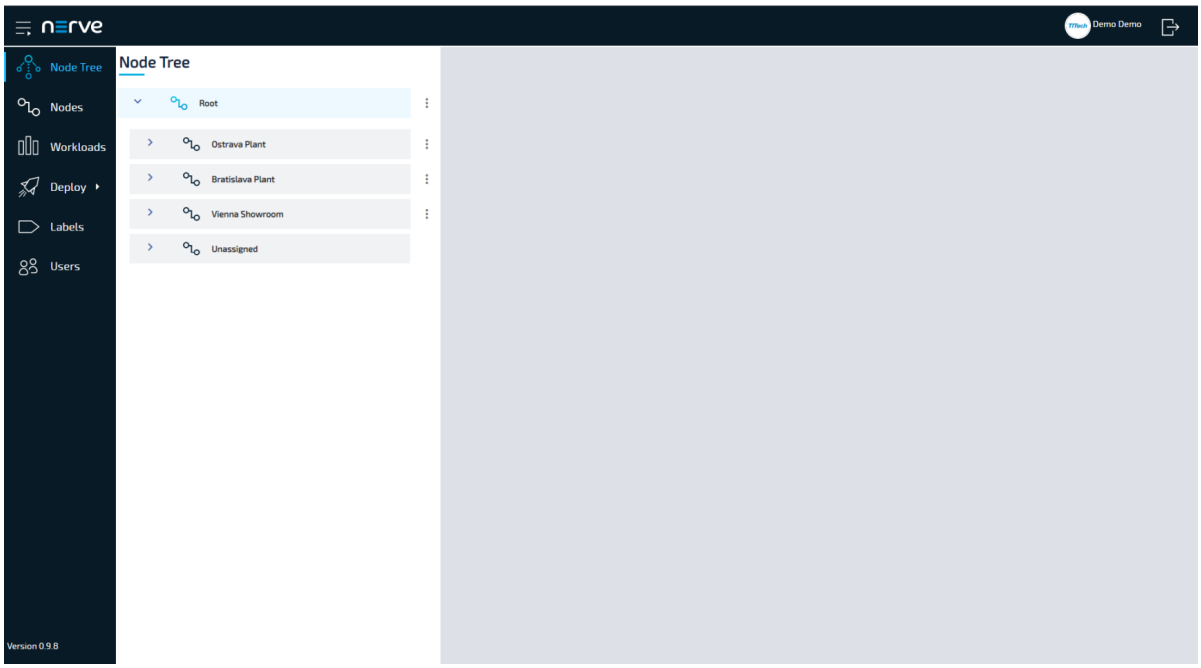
The command line tool offers many optional command line interface parameters. To see the list of parameters, enter python nerve-onboard.py -h. As described above, the command line tool will prompt you to define new passwords for the local UI, the CODESYS runtime and host access if no parameters are given. It will also ask for the password for Management System access. For the other parameters, the default values are used if they are not defined. The default parameters are given in the table below where applicable.

Item	Description
Usage	nerve-onboard.py [-h] [-v] [--mfn-ip IP] [--mfn-port <port>] [--user <username>] [--password <password>] [--current-rtvm-pwd <password>] [--new-rtvm-pwd <password>] [--new-system-pwd <password>] [--new-log-dest URL] [--new-localui-pwd <password>] [--cloud-user <username>] [--cloud-pwd <password>] [--name <nodename>] [--version <nodeversion>] [--label <labelkey>] <domain> <serial>
Positional arguments	<domain> This is the URL of your cloud. Please enter the URL without the protocol in the beginning. You can find it in the customer profile. Example: example.nerve.cloud <serial> Enter the serial number of your Nerve Device here. You can find it on the label of the Nerve Device.
Optional arguments	-h or --help Use this argument to show the help message in the command line. -v or --verbose Add this argument to show verbose output, giving detailed information during the registration process.
Node connection arguments	The following arguments are defined for connection to the node. --mfn-ip <ipaddress> This argument defines the IP address of the MFN 100. The default IP address is 172.20.2.1. --mfn-port <port> Define the port through which the local UI of the Nerve Device can be reached. The default port is 3000. --user <username> Enter the user for local UI login here. The default user is local@nerve.cloud. --password <password> Enter the current password for local UI login. The default is Passw0rd. --current-rtvm-pwd <password> Enter the current password to the CODESYS runtime here. The default password is lab123.

Item	Description
New configuration	The arguments here are used to define new passwords and paths. --new-rtvm-pwd <password> Define the new password for the CODESYS runtime. If this information is not given, you will be prompted by the command line tool. --new-system-pwd <password> Define the new password for SSH login to the host. If this information is not given, you will be prompted by the command line tool. --new-log-dest <url> Define the path for the elasticsearch log destination. The default path is <domain>:443/filebeat_app/. Please note that logging should be available in the Management System by default. --new-localui-pwd <password> Define the new password for local UI login. If this information is not given, you will be prompted by the command line tool.
Cloud connection	The following arguments are used for connecting to the cloud. The username and password here are also used for the web interface. --cloud-user <username> Enter the username to connect to the cloud. The default user is admin@nerve.cloud. --cloud-pwd <password> Enter the password to connect to the cloud. If this information is not given, you will be prompted by the command line tool.
Node details	You can configure the details for how this node will appear in the cloud. The information defined here can be configured in the Management System as well after node registration was successful. --name <nodename> This is the name of the node that will be displayed in the Management System. The default name is the serial number of the node. --version <nodeversion> Define a node version. The default node version is 2.0.0. --label <key> <value> Add a label to the node. Define a key for the label which function as the "category" of the label (e.g. MachineNumber) and a corresponding value (e.g. 1). This can be given multiple times to add multiple labels. However, please note that a node can only be assigned one label per label key. Label keys must consist of one word only.

Node Tree

The Node Tree is the first screen you will see after logging in to the Management System. It presents an overview of all nodes that are connected to the Management System, embedded into tree view elements. Being mainly a means of organization, it has no impact on the functionality of the nodes and their workloads.

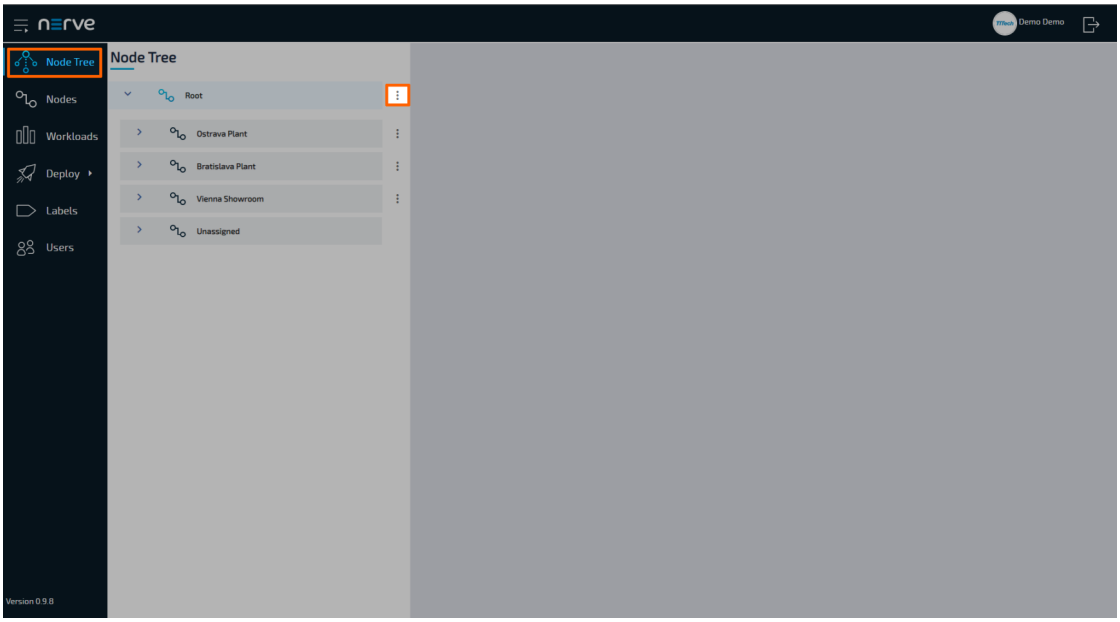


There is only one element under the root after the initial setup: **Root > Unassigned**. All nodes that are registered in the Management System are placed in the **Unassigned** element by default. From there they can be moved to new elements that have to be created first.

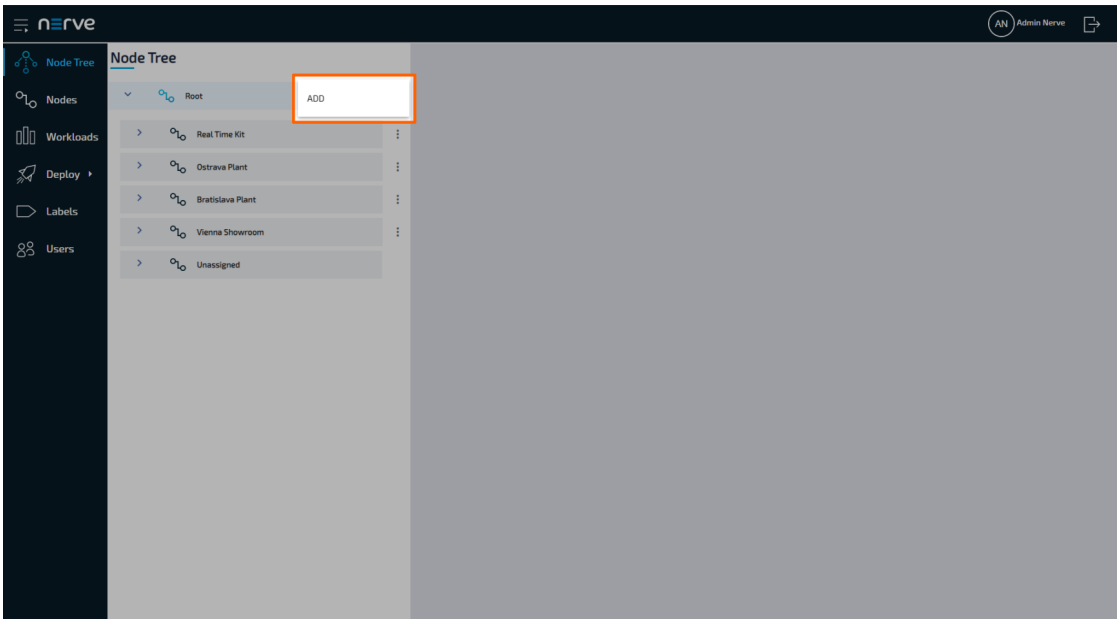
Creating a New Element in the Node Tree

Before you can move a node out of the **Unassigned** element, you have to create a new element in the Node Tree. Elements in the Node Tree exclusively serve an organizational purpose.

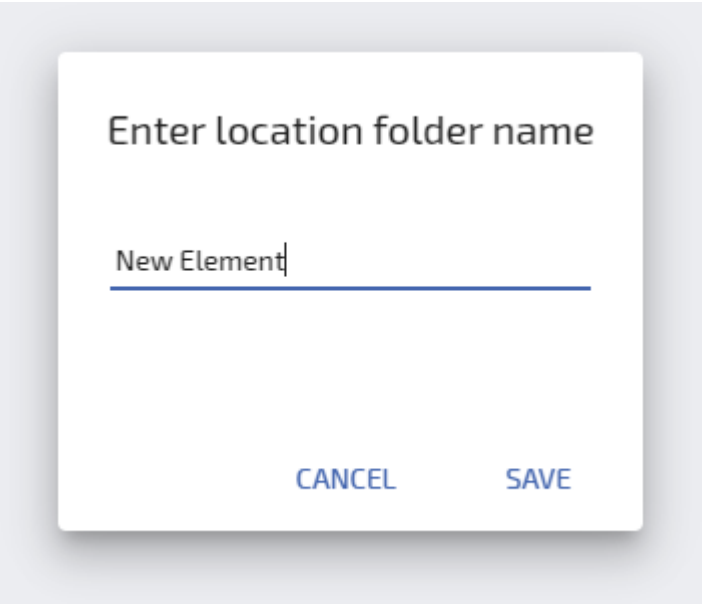
1. Select **Node Tree** in the left-hand menu.
2. Select the ellipsis menu to the right of **Root** in the Node Tree.



3. Click **Add** in the overlay that popped up.

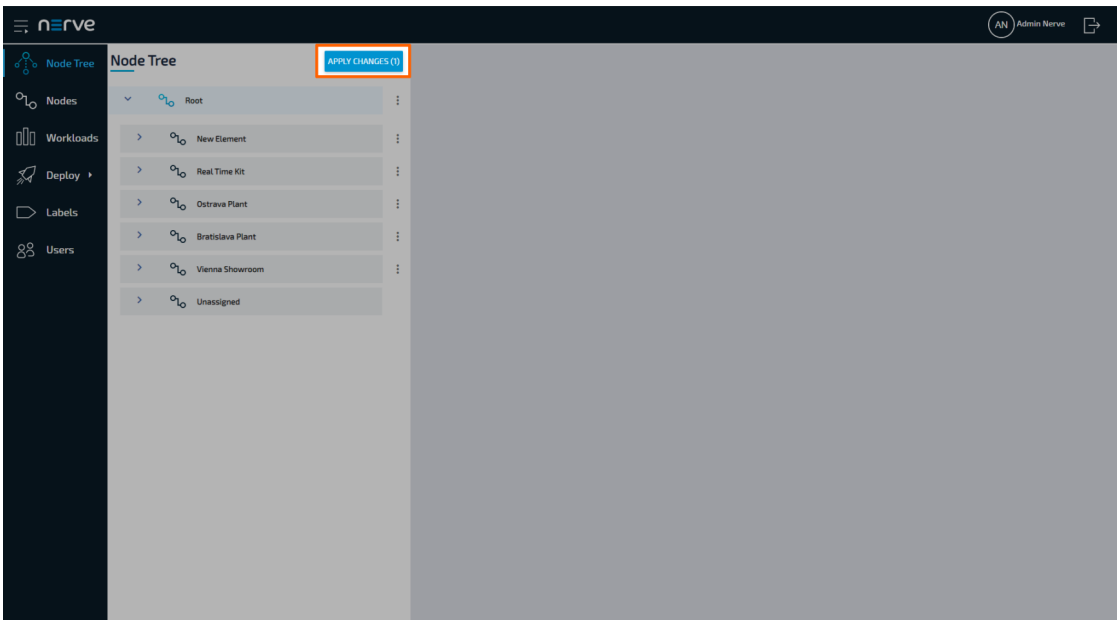


4. Enter the name of the new element under **Location name**.



5. Click **Save**.

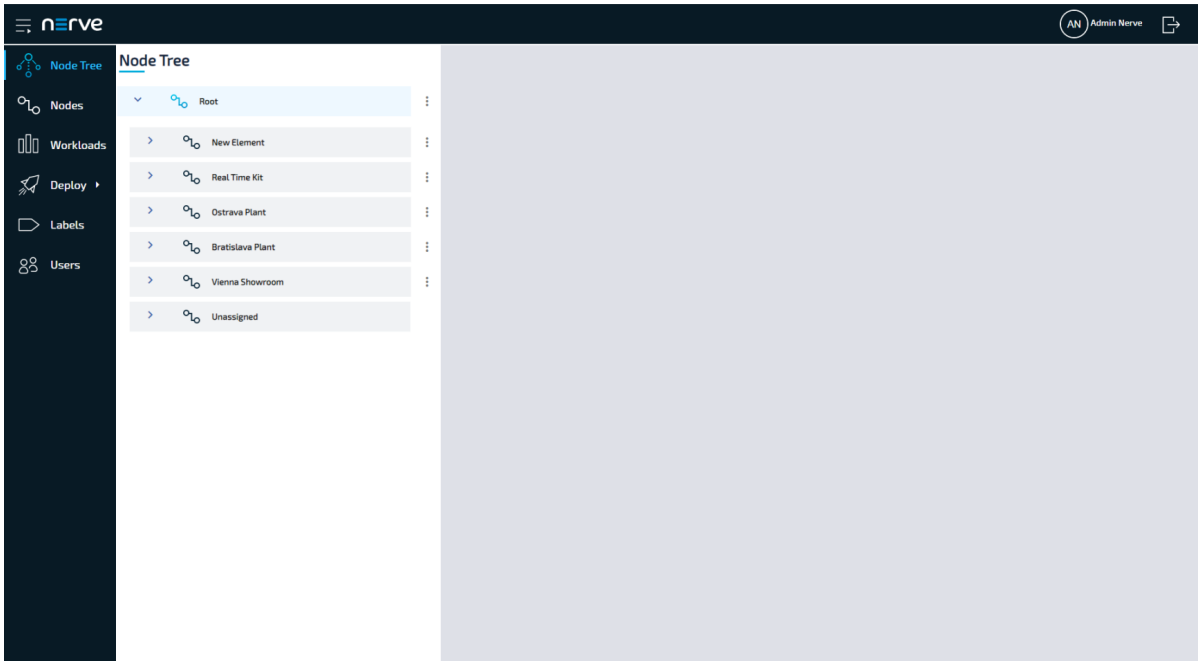
6. Select **APPLY CHANGES (n)** in the upper-right corner of the Node Tree.



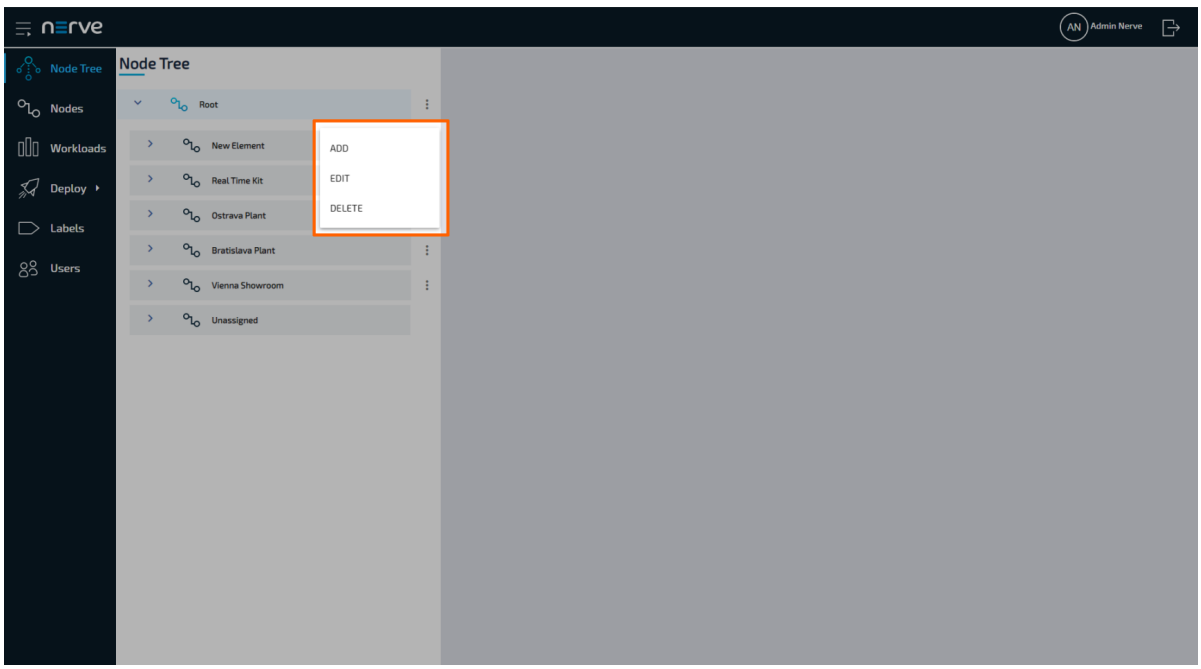
NOTE

(n) is a placeholder for the number of changes made to the Node Tree. If you have performed three changes, you will see (3) in the button above the Node Tree.

The new element now appears under the **Root** element.



You can create more elements and modify the Node Tree to your liking. To the right of every created element, you can see an ellipsis menu that opens up an overlay. Here you can add additional elements below the new element, edit the name of the element or delete the new element.



NOTE

- The order of the tree elements can be modified easily. You can just drag and drop the elements to the position you wish.
- When a tree element is deleted, all of the nodes inside the element will be moved to **Unassigned**.

Moving a Node from One Tree Element to Another

Moving nodes in the Node Tree is very straightforward and intuitive and possible by drag and drop. Please make sure that you have created a new tree element before attempting to move a node.

1. Select **Node Tree** in the left-hand menu.
2. Expand the tree element of the node you want to move. The default element is **Root > Unassigned**.
3. Choose the node you want to move.
4. Drag and drop the node to the newly created element. Elements expand automatically once you hover over them.

5. Select **APPLY CHANGES (n)** in the upper-right corner of the Node Tree.

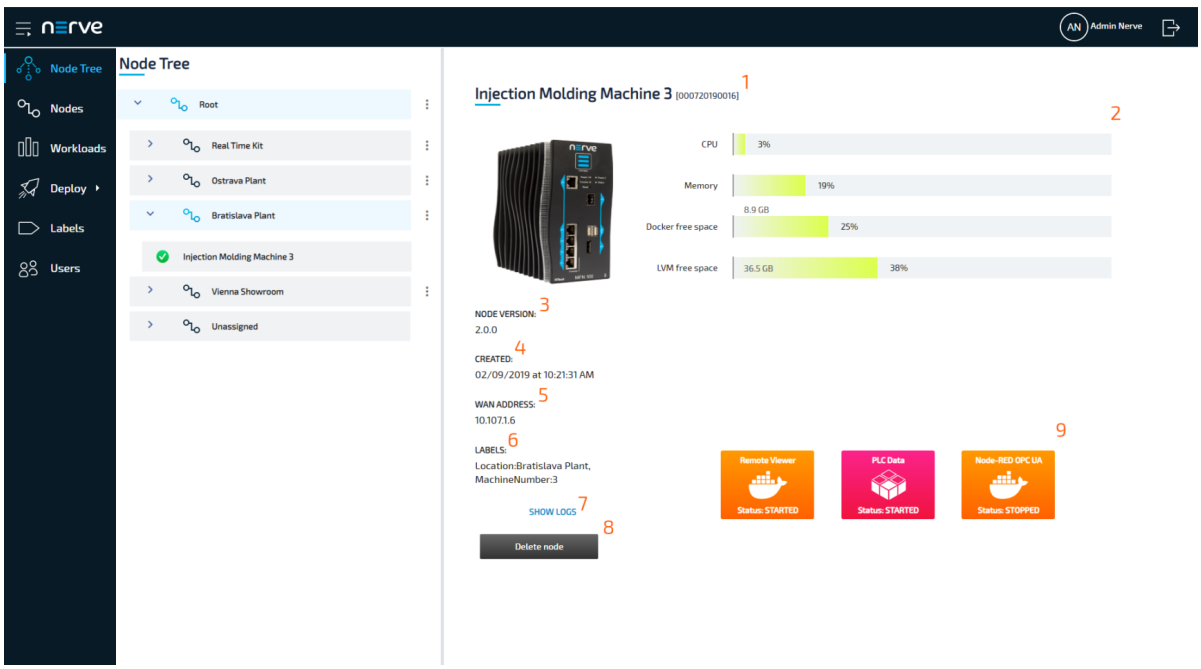
> **NOTE**
>
> (n) is a placeholder for the number of changes made to the Node Tree. If you have performed three changes, you will see (3) in the button above the Node Tree.

The node has now been moved to the new element.

> **NOTE**
>
> A node cannot be moved back to **Unassigned** once it has been moved to another element.

Managing Nodes in the Node Tree

Once you have nodes connected to the Management System, you can view their details next to the Node Tree. To view the details of a node, choose a node and click the node name or symbol.



Item	Description
Node name and serial (1)	Here the name and serial number of the node are shown. The serial number is next to the name in brackets. Please note that the serial number of the node is its default name if you have not defined a node name in the registration process. Also, depending on your Nerve Device, the command line tool might extend the serial number to 12 characters. Please make sure to take note of that after registering a Nerve Device.
System metrics (2)	The system metrics that are available in the local UI dashboard are also visible here: • **CPU** The percentage here shows how much processing power is being used in total at the moment. This includes CPUs that have been assigned to VMs and Docker Containers as well. • **Memory** Similar to CPU usage, the percentage of memory used in total at the moment is shown here. This includes memory that has been assigned to VMs or Docker Containers. • **Docker free space** Two things are shown in this graph: The percentage shows how much of the available space for Docker containers is already used. The value shows the amount of space that is free. Docker containers have their dedicated virtual partition. • **LVM free space** Similar to Docker free space, two things are shown in this graph as well: The percentage shows how much of the available space for virtual machines is already used. The value shows the amount of free space. Please note that deployed Virtual Machine workloads share a logical volume group with the Nerve Blue Base System. Therefore the percentages and values displayed in this graph are in relation to the total amount of space available of the logical volume group.
NODE VERSION (3)	The version of the node. This is set to 2.0.0 by default if the node version has not been set in the registration process.
CREATED (4)	This is the date when the node was added to the Management System.
WAN ADDRESS (5)	This is the network address of the node that has been assigned by the DHCP server.
LABELS (6)	Here you can see labels that are assigned to this node. Labels can be set in the registration process and the Management System. See Labels for more information.

Item	Description
SHOW LOGS (7)	Clicking here will open a new window and show the system logs of the node. The logs are visualized with the Kibana application. However, please note that the logs shown are unfiltered and therefore require expert knowledge due to their number and contents. If clicking here does not show logs in the Kibana application, please contact TTTech Industrial customer support.
Delete node (8)	Clicking here removes the node from the Management System. The node needs to be registered again after it has been removed.
Workloads overview (9)	Here you will find workloads that have been deployed to the Nerve Device displayed in tiles. Clicking these tiles leads to a control area in which you can control the workload. If there are no tiles, no workloads have been deployed to the Nerve Device.

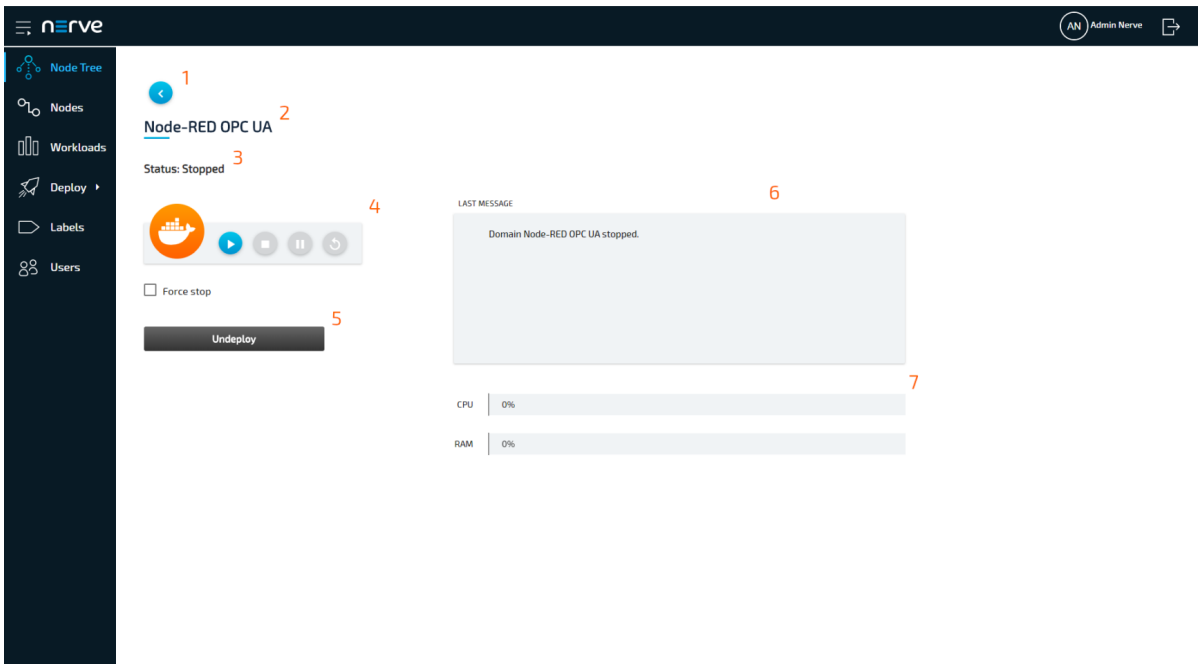
Workload Control

All workloads that have been deployed to the node are displayed in tiles below the node details in the Node Tree. Clicking these tiles allows you to control the respective workload.

NOTE

CODESYS workloads can only be controlled from the local UI.

The workload control screen offers the same options for both Virtual Machine workloads and Docker workloads.



Item	Description
Back button (1)	Click here to return to the Node Tree.
Workload name (2)	This is the name of the workload. The name of the workload version is not displayed here.
Workload status (3)	The current status of the workload is displayed here. The possible statuses are the following: • **Idle** This is the initial state of the workload before it is started. • **Creating** This is a transitional state of the workload when it is being created on the node. • **Starting** This is a transitional state when the workload is being started. • **Restarting** This is a transitional state when the workload is being restarted. • **Started** The workload is running and operating. • **Suspending** This is a transitional state when the workload is being suspended. • **Suspended** The workload has been paused. • **Resuming** This is a transitional state when the workload is being resumed from the suspended state. • **Stopping** This is a transitional state when the workload is being stopped. • **Stopped** The workload has been stopped. • **Removing** This is a transitional state when the workload is being undeployed. • **Error** An unknown error has occured.
Control panel (4)	There are five control options for workloads here: • **Play** If the workload is in a stopped state, clicking **Play** will start the workload. • **Stop** If the workload is running, clicking **Stop** will stop the workload. • **Suspend** Clicking **Suspend** will pause the workload. It can be continued by clicking **Play**. • **Restart** This will restart the workload. • **Force stop** Setting a value here will force the workload into the stopped state once **Stop** is clicked and the set time has expired.

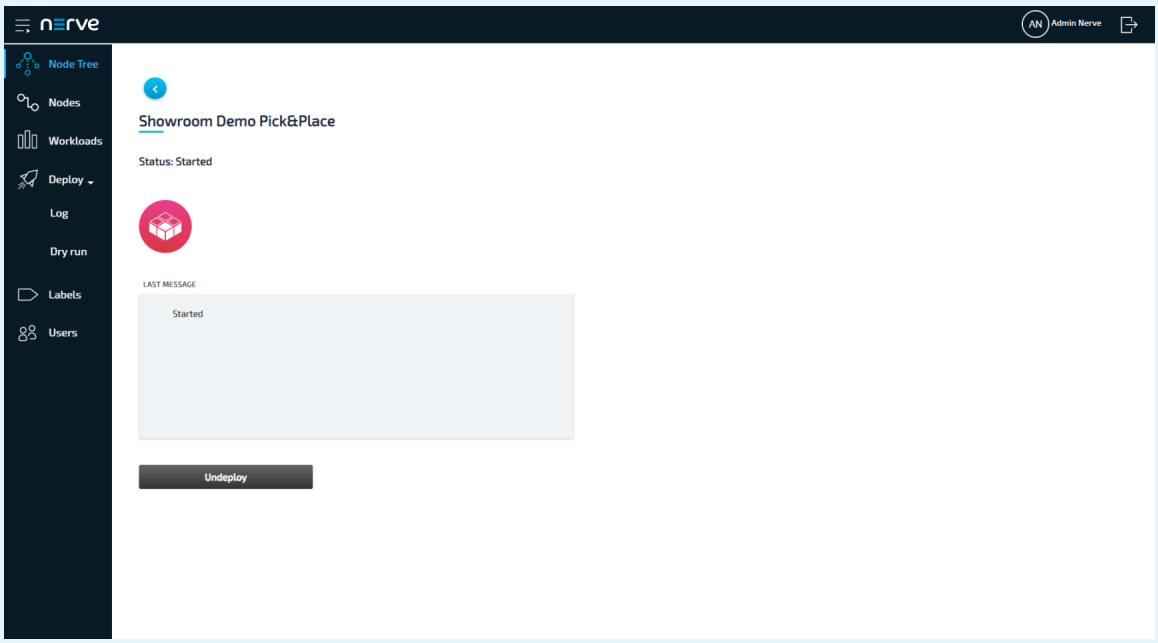
Item	Description
Undeploy (5)	Clicking this removes the workload from the node. The tile in the node detail screen disappears. The workload stays in the list of workloads in the Management System and can be deployed again at any time.

Item	Description
Message window (6)	The message window displays the latest message the workload has sent out. The type of message that is displayed here depends on the workload. Here is a list of messages that are valid for VMs and Docker containers: • "Domain creating." • "ERROR during creating! <errormessage>" • "Domain starting." • "ERROR during starting! <errormessage>" • "Domain <domainname> started." • "Domain stopping." • "ERROR during stopping! <errormessage>" • "Domain <domainname> stopped." • "Domain suspending." • "ERROR during suspending! <errormessage>" • "Domain <domainname> suspended." • "Domain resuming." • "ERROR during resuming! <errormessage>" • "Domain restarting." • "ERROR during restarting." • "Domain removing!!!" • "ERROR during removing." • "ERROR!!! Domain stopping." In the messages above, <domainname> is a placeholder for the name of the VM or Docker. In case of Docker containers, <errormessage> signifies a message that is generated by the Docker container if an error occurs. For VMs, there is an additional set of messages: • "Failed to connect to hypervisor." • "Failed to create domain." • "Domain <domainname> created." • "Cannot start <domainname> domain because it may already be running!" • "Failed to resume <domainname> domain!" <errormessage> • "Failed to start domain <domainname>. " <errormessage> In this case, <errormessage> is a message that is fetched from the libvirt library. CODESYS workloads have the following set of messages: • "Preparing files for installation" • "Starting CODESYS application" • "CODESYS application started" • "Stopping CODESYS application" • "CODESYS application stoppped" • "Removing CODESYS application file" • "An unexpected error has occurred. <errormessage>" Here, <errormessage> is a message that is sent between the node and CODESYS.

Item	Description
Usage statistics (7)	Virtual Machine workloads and Docker workloads have their assigned resources they can use. The use of these resources is displayed with bar graphs: • **CPU (VM and Docker)** The percentage here shows the usage of CPU resources in relation to the assigned CPUs. Example: A VM is assigned one CPU core out of four and the core is at 75 % usage capacity. The graph will be at 75 %. • **RAM (Docker only)** Similar to the CPU usage statistic, the percentage here shows the usage of system memory resources in relation to the assigned memory. If the assigned memory is at a 100 % usage capacity, the graph will be at 100 %. If no memory has been assigned, the graph will show the percentage of used memory in relation to the total available memory of the host.

NOTE

Since CODESYS workloads can only be controlled through the local UI, the workload control screen does not offer any control options. It offers a message window and the option to undeploy the workload.

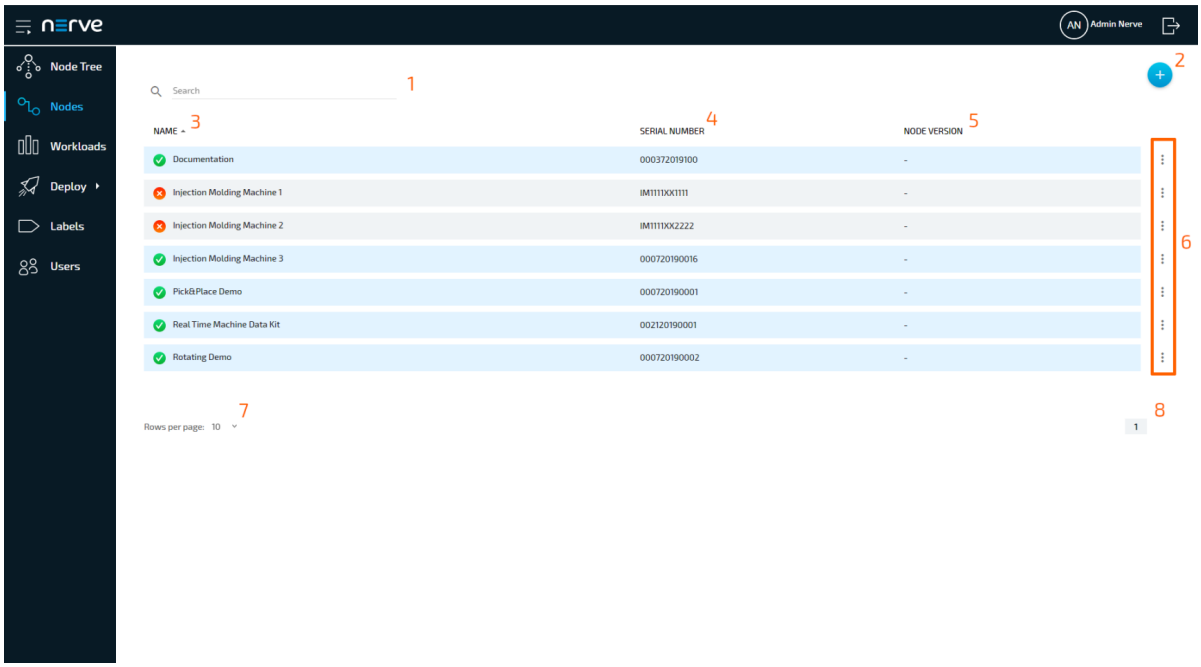


Nodes

This chapter will walk you through node management and basic operations in the UI. The topics covered in this chapter are mainly means of organization and have no impact on the functionality of the nodes and their workloads.

NOTE

Operational functions of the nodes are located in the node details view in the Node Tree.



Item	Description
Search bar (1)	Use the search bar to filter nodes by name.
Add new node (2)	Click here to manually add a new node.
NAME (3)	This is the name of the node. If a node is online or offline can be seen to the left of the name. A green check mark indicates an online node while a red cross shows an offline node. The sorting of the list can also be switched from alphabetical to reverse alphabetical by clicking **NAME**.
SERIAL NUMBER (4)	This is the serial number of the node that was entered during the node registration process. It was possibly extended to 12 characters by the registration tool.
NODE VERSION (5)	This is the version of the node. The default value is 2.0.0.
Ellipsis menu (6)	Clicking here opens an overlay that allows deleting nodes.
Rows per page (7)	Specify how many nodes are displayed on one page. You can select **5**, **10** or **15** nodes per page.
Page navigation (8)	Use the arrows to switch between pages. Clicking the number in the middle opens a list of all page numbers. Selecting a number jumps to that page.

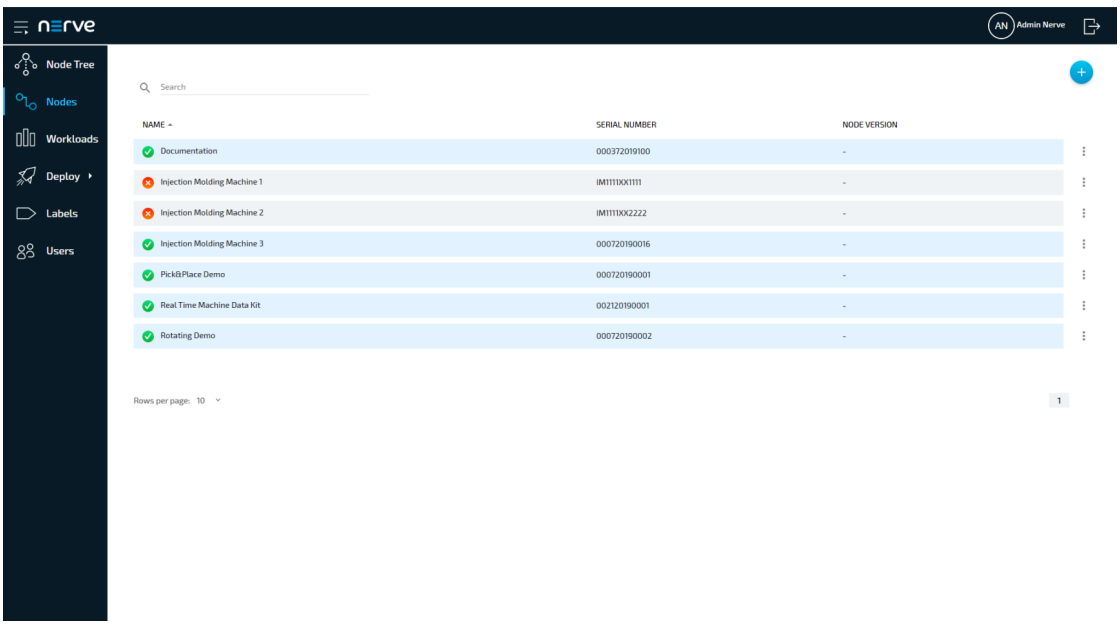
Adding a Node Manually

NOTE

If you are adding a new node, please refer to the node registration chapter.

In this version, there is the possibility to manually add nodes to the Management System. This way of adding nodes is only possible if a node has been added previously through the registration process and if its details are known.

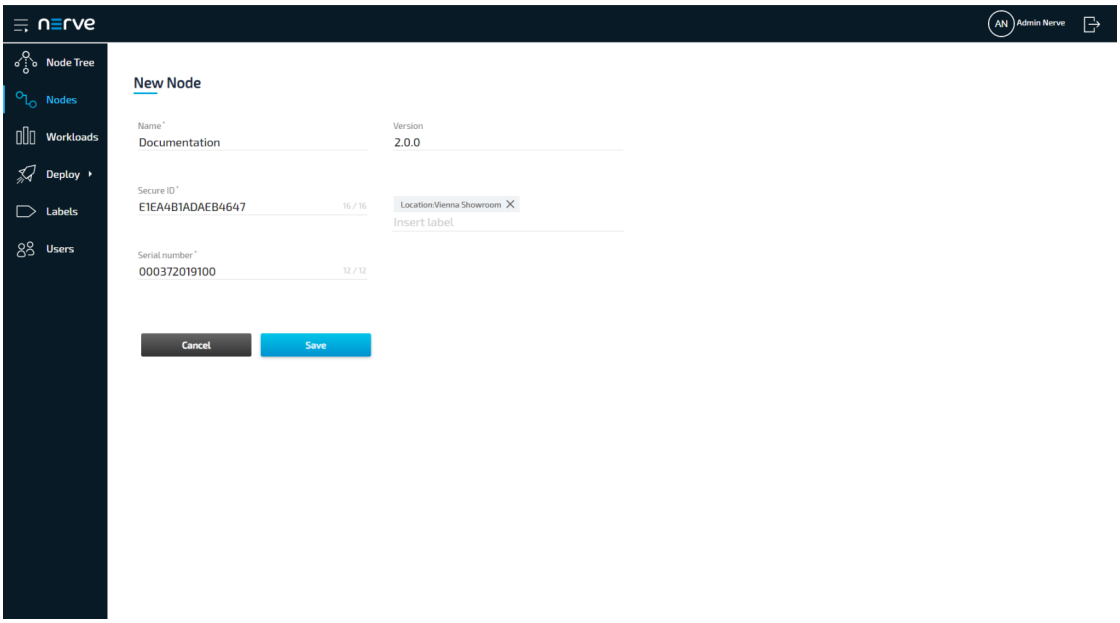
1. Select **Nodes** in the menu on the left side.
2. Select **Add new node** icon in the upper-right corner.



3. Enter the following information:

Item	Description
Name	Enter the name for the node that will help you identify it easily.
Secure ID	This ID is generated when the node is registered in the Management System. There is no way to generate and obtain this ID other than registering the node with the registration tool.
Serial no.	This is the serial number of your Nerve Device. Please note that the serial number needs to have 12 characters. If the serial number of your Nerve Device does not have 12 characters, please extend the serial number.
Version	This field is optional. You can define a node version here as a means to help with organization. The default value set by the registration tool is 2.0.0.
Insert Label	This field is optional. You can add labels to the node for easier identification and workload deployment.

4. Click **Save** to save your changes.



The node now appears in the node list and in **Root > Unassigned** in the Node Tree.

> **NOTE**
>
> Labels have to be created first before they can be assigned to nodes. See Labels for information on how to create labels.

Editing the Details of Nodes

After registering a node through the registration process, you can edit its details again in the **Nodes** menu. If you have a large number of nodes, you can use the search bar at the top to search for nodes.

1. Select **Nodes** in the left-hand menu.
2. Click a node from the list.
3. Edit the details of the node:

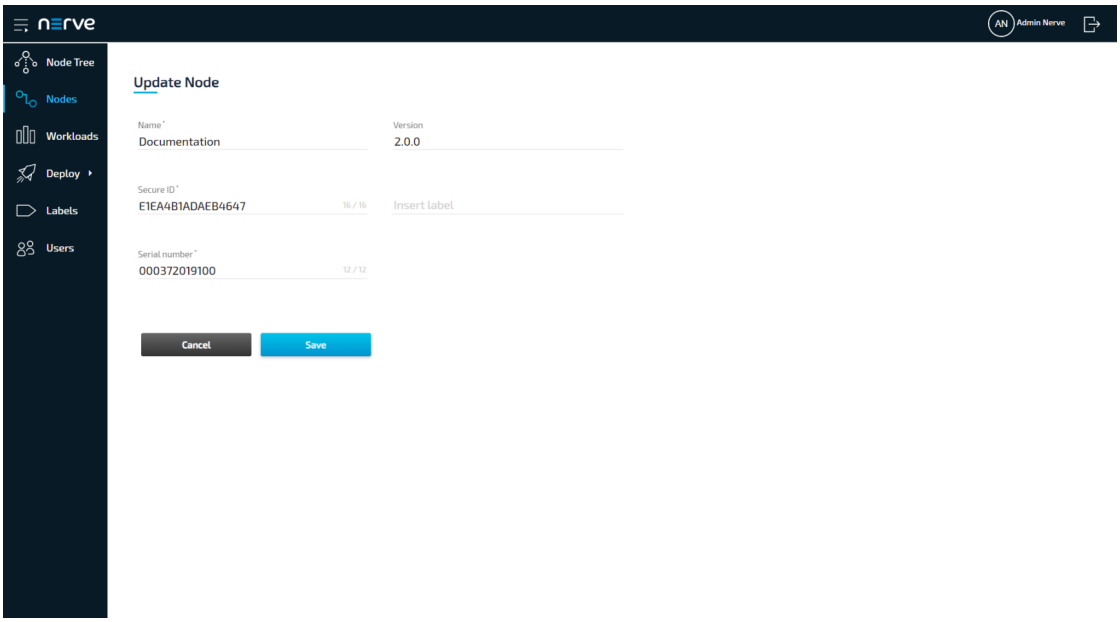
Item	Description
Name	Enter the name of the node. The default name of a node is its serial number.
Secure ID	This is the ID generated during the node registration process. Do not edit this ID without a valid reason.
Serial number	This is the serial number of the node. Do not edit the serial number without a valid reason.
Version	This is the node version. The default value is **2.0.0**.

Item	Description
Insert label	If labels have been defined in the Management System, assign them to nodes here. Start typing to see a list of available labels in a drop-down menu. Please note that only one label per label key can be assigned.

NOTE

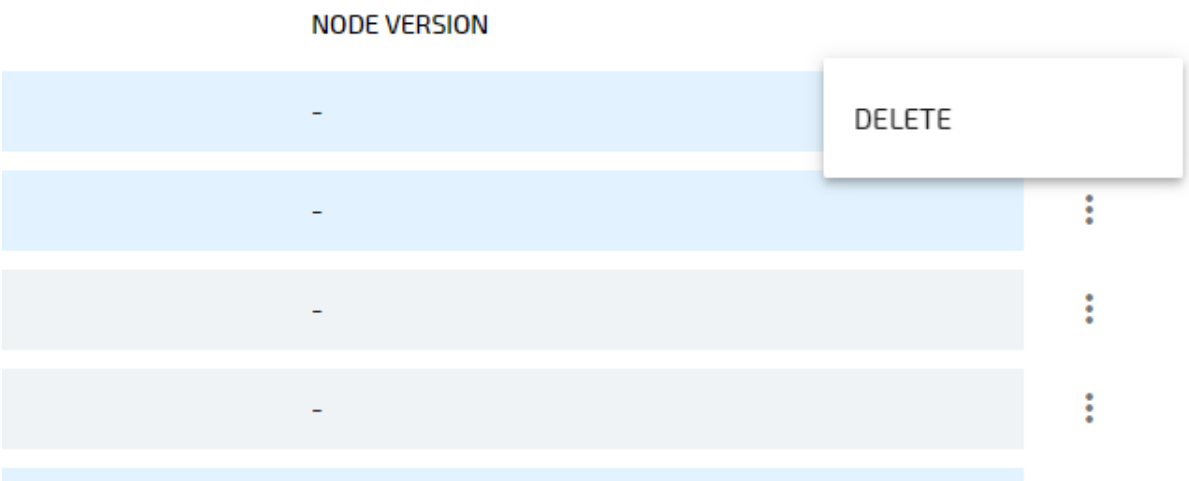
Changing the **Secure ID** or **Serial no.** of a node will break the connection between the node and the Management System.

4. Click **Save** to save your changes.



Removing a Node

Please note that generally there is no need in removing a node. Only remove a node in case of technical difficulties or if customer support instructs you to. If you need to remove a node, select **DELETE** in the ellipsis menu on the right side of a node in the node list.



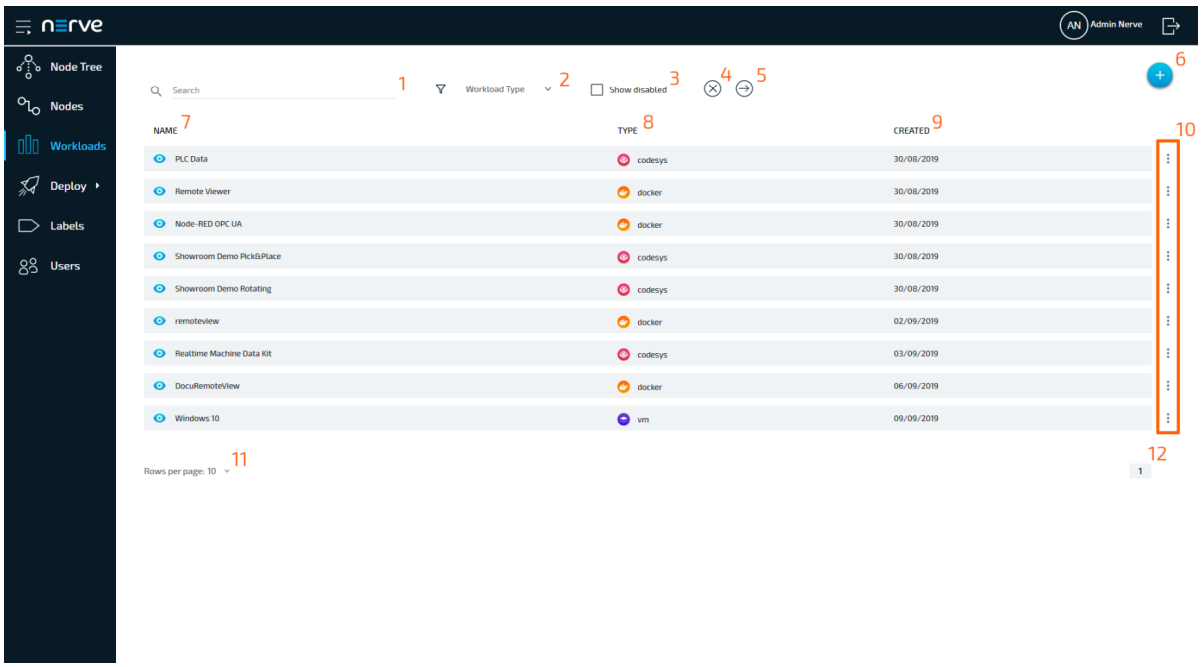
If you need to register the node in the Management System again, you will need to follow the node registration process

Workloads

In order to work with CODESYS applications, virtual machines or Docker containers on nodes, workloads need to be provisioned in the Management System. Here, provisioning is the creation of a workload and its storage in the workload repository of the Management System so that it can be deployed to nodes. This requires configuration of the workload and files that need to be uploaded to the Management System. After that, the workload can be deployed to nodes.

There are three types of workloads that can be provisioned: CODESYS workloads, Virtual Machine workloads and Docker workloads. The process of provisioning each workload is described in their respective chapters.

Select **Workloads** in the left-hand menu to find a list of all workloads that have been provisioned.



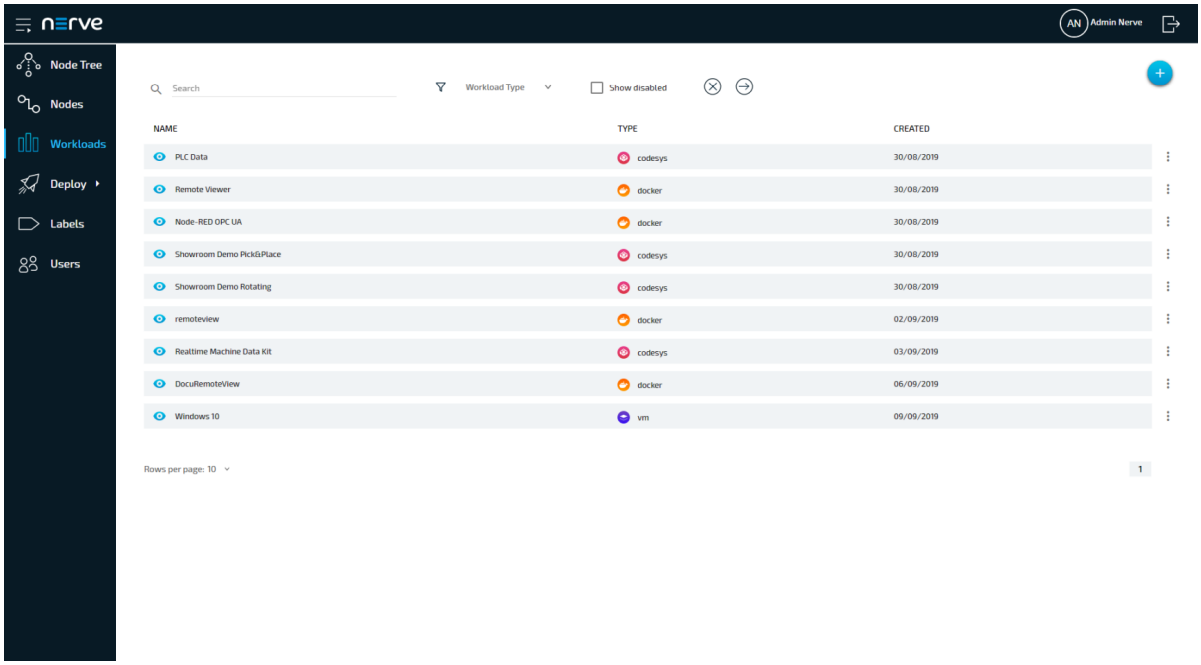
Item	Description
Search bar (1)	Use the search bar to filter workloads in the list by name.
Workload Type (2)	This is a drop-down menu that allows you to filter the list below by workload type. The available options are **VM**, **Docker**, **CODESYS** and **All**.
Show disabled (3)	Disabled workloads are not shown in the list of workloads. Ticking this checkbox shows them again. However, please note that this does not enable the workloads again.
Clear search query (4)	Clicking the cross symbol resets the search query to its initial values.
Execute search (5)	Clicking here executes the search according to the search criteria you have defined. Pressing Enter also executes a search.
Add new workload (6)	Click the plus symbol to provision a new workload.
NAME (7)	This is the name of the workload that has been defined in the provisioning process.
TYPE (8)	The type of workload is displayed here: **codesys**, **vm** or **docker**.
CREATED (9)	This is the date when the workload was provisioned in the format DD/MM/YYYY.
Ellipsis menu (10)	Clicking here opens an overlay that gives two options: **DELETE** and **DISABLE**.
Rows per page (11)	Specify how many workloads are displayed on one page. You can select **5**, **10** or **15** workloads per page.
Page navigation (12)	Use the arrows to switch between pages. Clicking the number in the middle opens a list of all page numbers. Selecting a number jumps to that page.

Provisioning a Workload

Provisioning a workload is the creation of a workload in the workload repository of the Management System. Workloads that have been provisioned are ready to be deployed to nodes. Select the plus symbol in the upper-right corner of the workloads list to start provisioning a workload. The provisioning process of each workload type is covered separately in the following chapters.

- Provisioning a CODESYS Workload
- Provisioning a Virtual Machine Workload
- Provisioning a Docker Workload

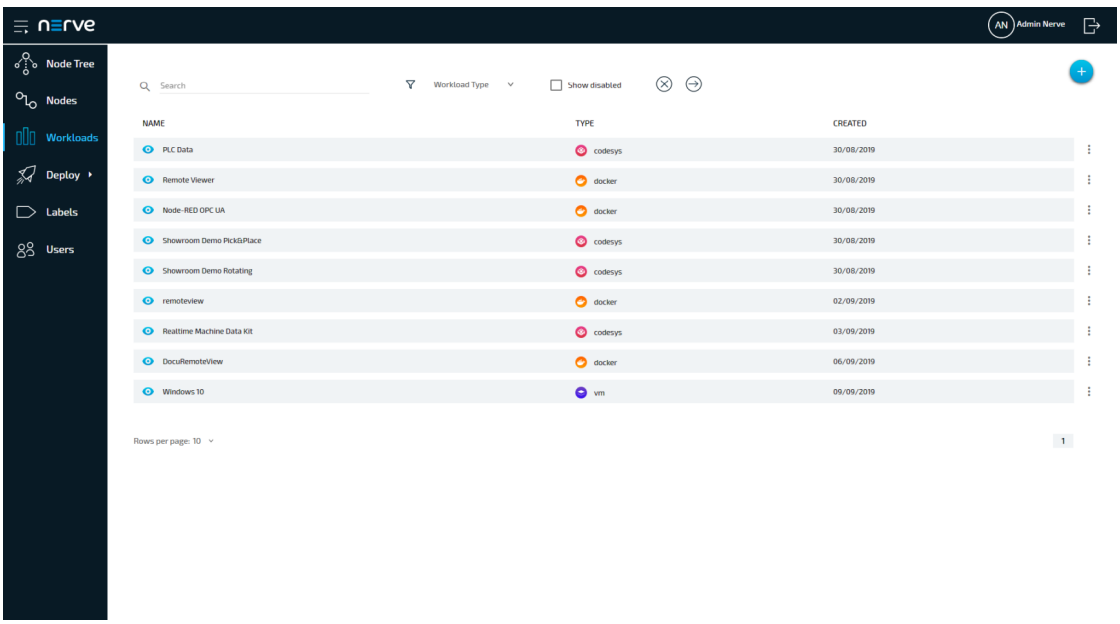
After you have provisioned a workload, it will appear in the list of workloads.



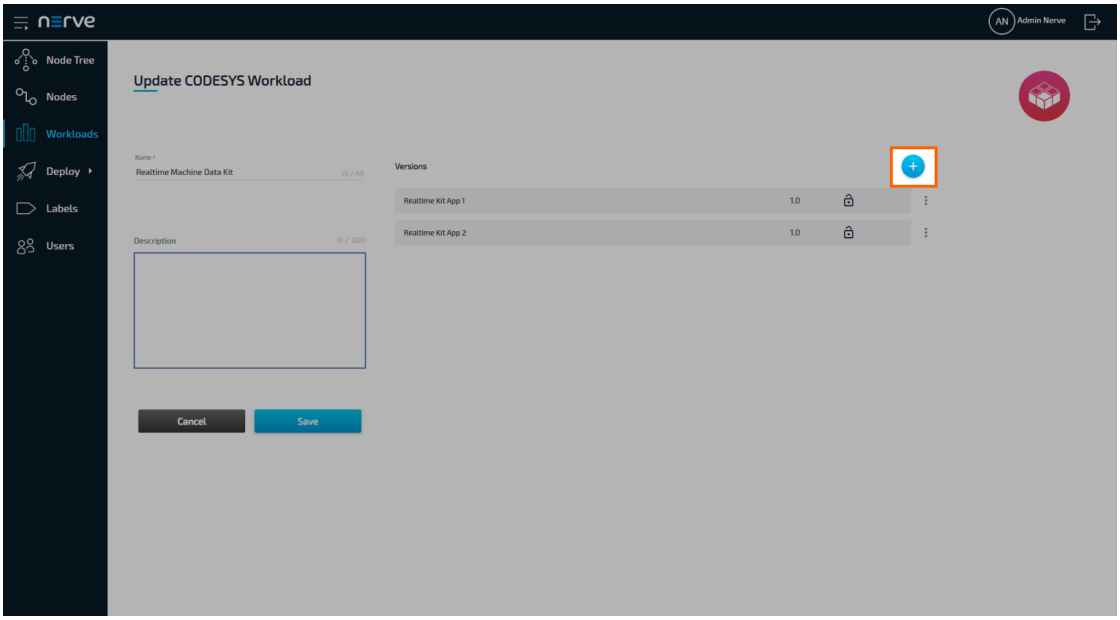
Adding a New Workload Version

You can add new versions to a provisioned workload to accommodate different use cases.

1. Select **Workloads** from the menu on the left side.
2. Select the workload to which you would like to add a new version.

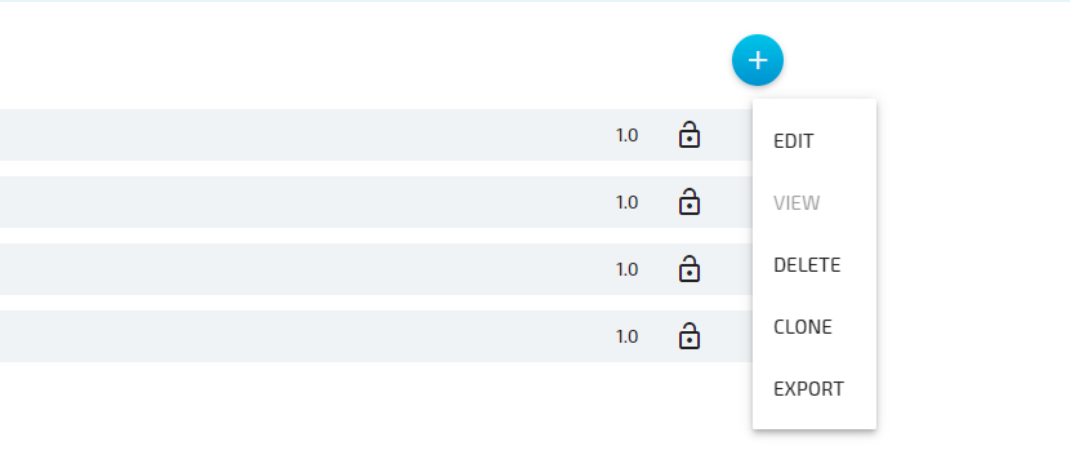


3. Click the plus symbol in the upper-right.



NOTE

If you want to add a new version that is a slight modification of an existing one, click the ellipsis menu next to a workload and select **CLONE** from the overlay.

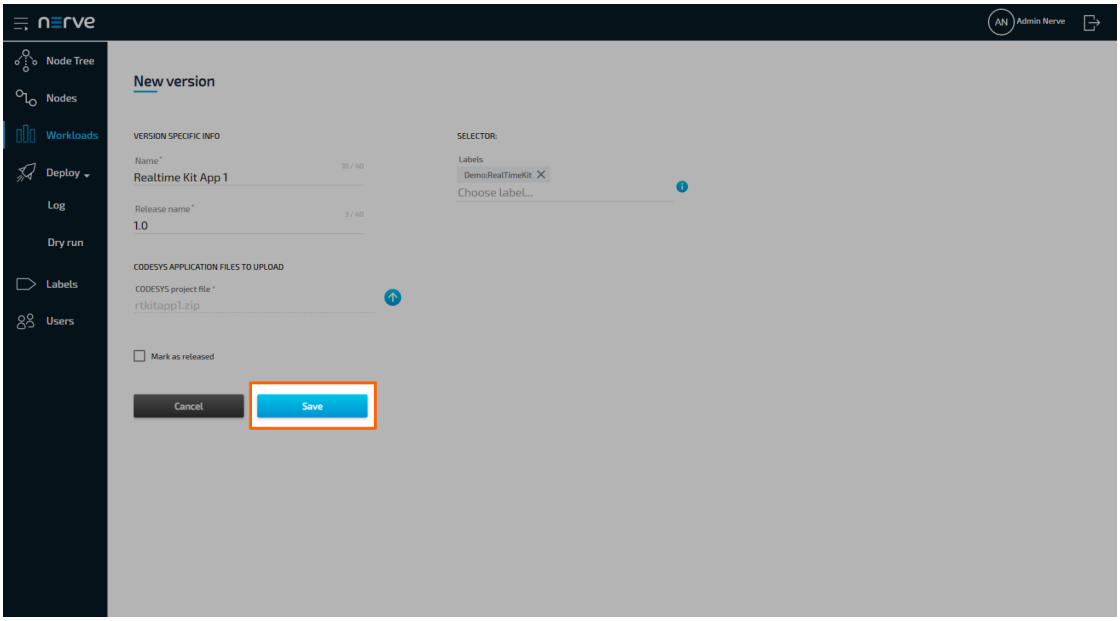


4. Configure the new workload version.

NOTE

The fields of the new version will already have information filled in. The system enters the settings of the latest version automatically. If you clicked **CLONE** next to a workload version, the information filled in will be from that version instead.

5. Click **Save** to save the new version of the workload.

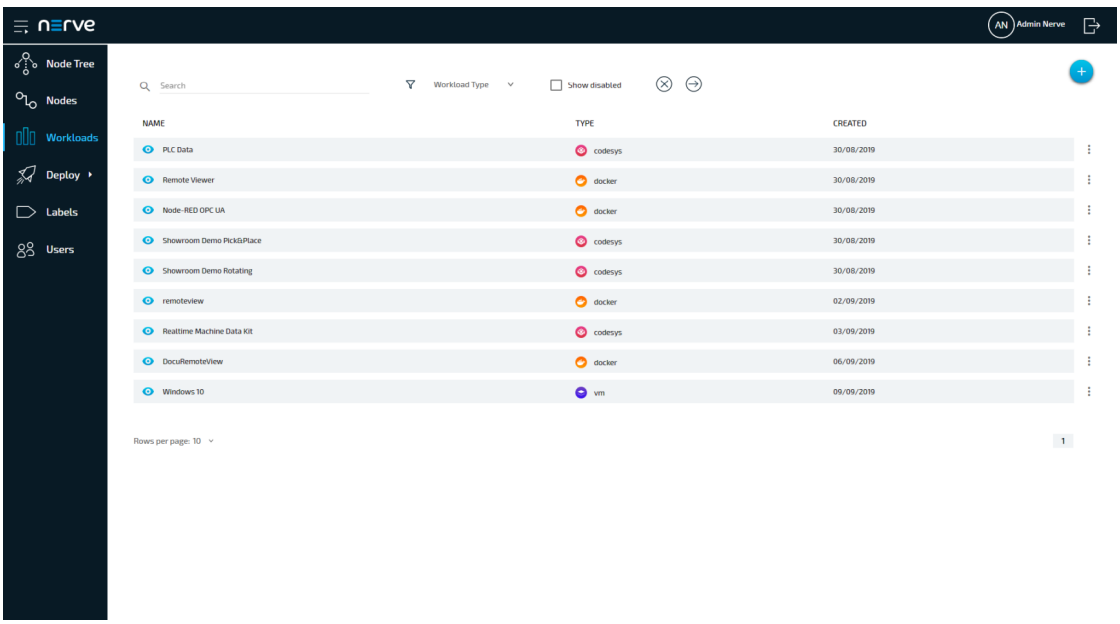


Editing a Workload

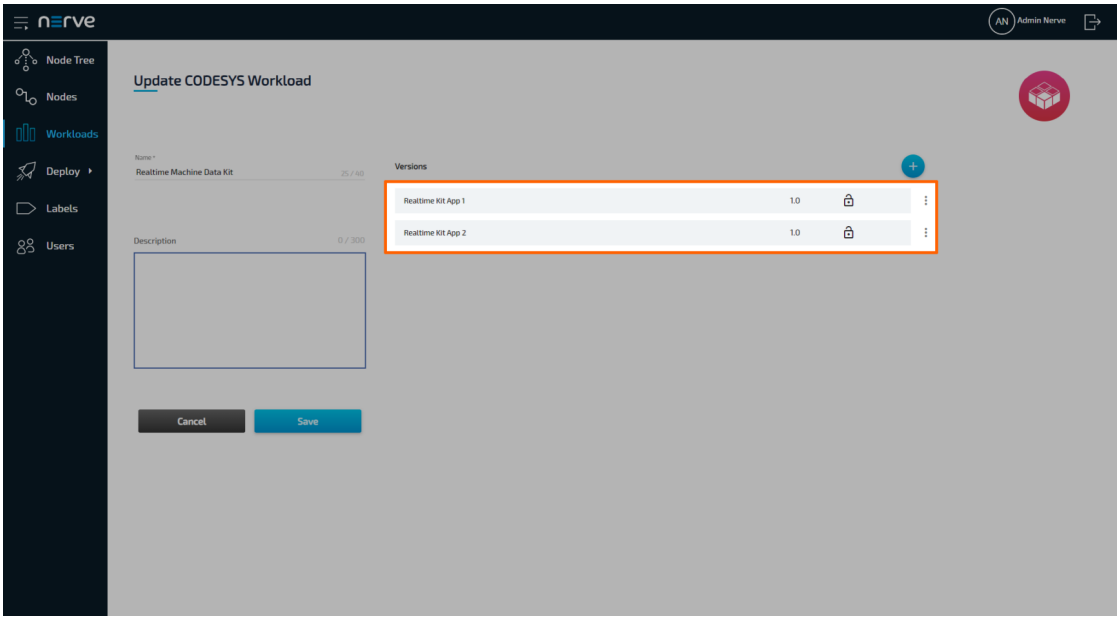
General information of a workload and configuration settings of each version can be edited starting from the workload list. General information of a workload is valid for all workload versions.

1. Select **Workloads** from the menu on the left side.

2. Select the workload you would like to edit.

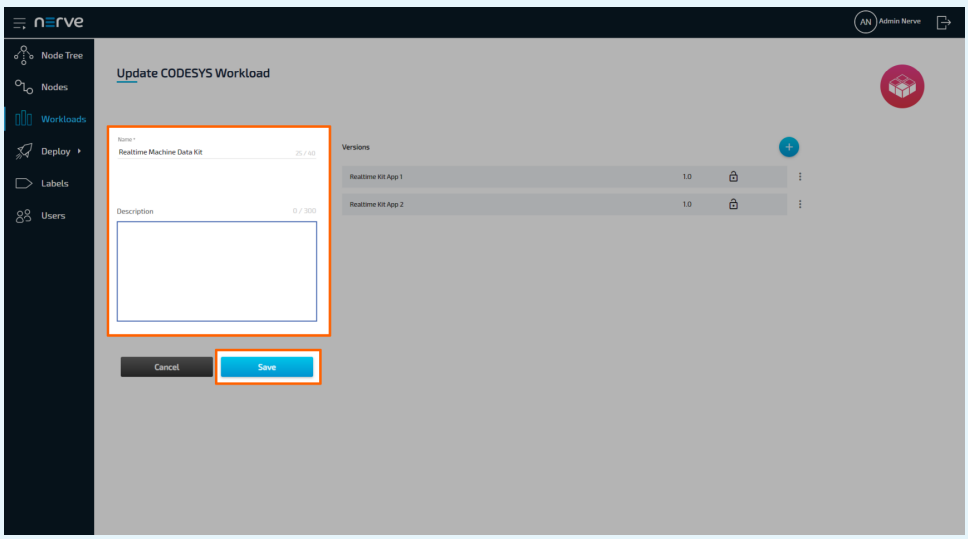


3. Select the workload version you would like to edit from the list on the right.



NOTE

Editing the **Name** and **Description** of the workload can be done on the left side before selecting a version. Perform the desired changes and click **Save**.

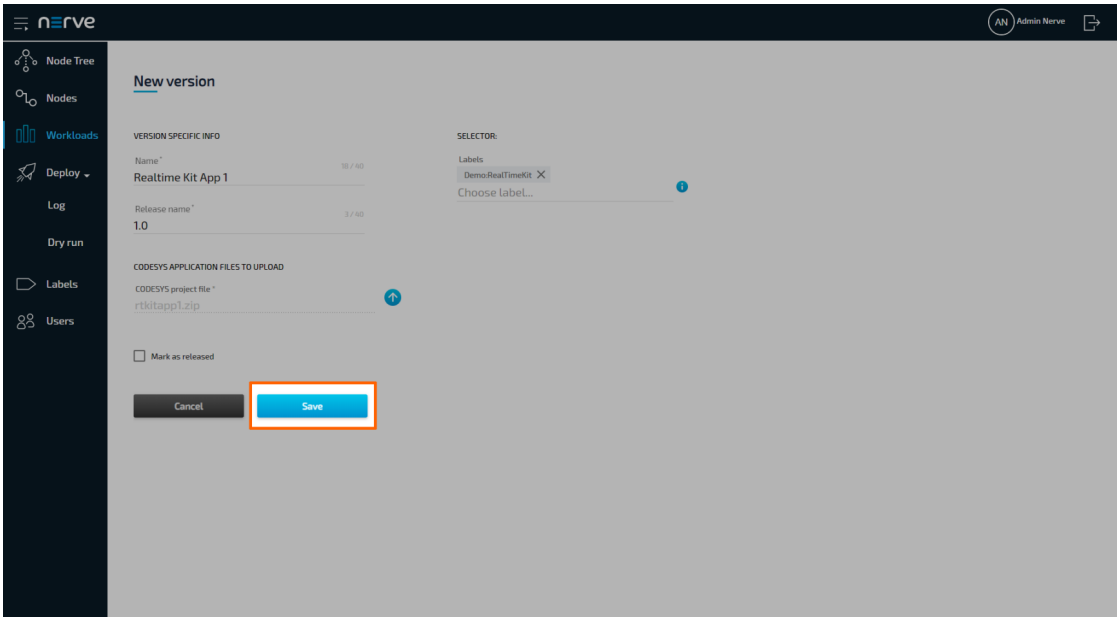


4. Perform your changes to the workload version.

NOTE

The settings of a workload depend on the workload type. See the version settings for each workload in the provisioning chapters: CODESYS workloads, Virtual Machine workloads and Docker workloads.

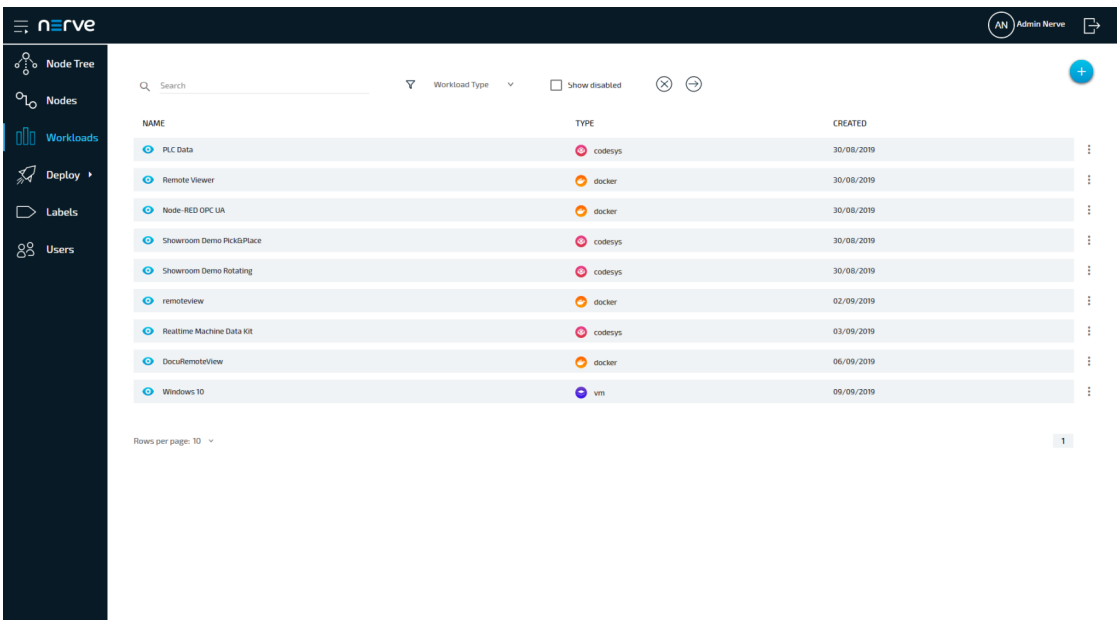
5. Click **Save** to save your changes.



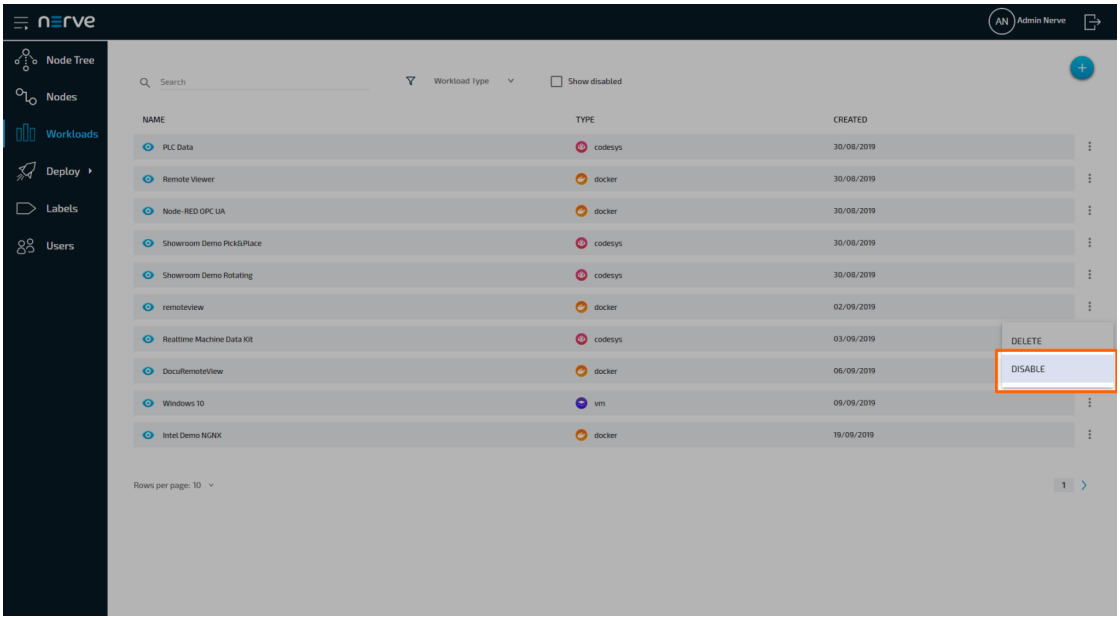
Disabling a Workload

A workload can be disabled to make it hidden and not selectable. This will hide the workload in the workload list and deployment process but it will not be deleted from the Management System. This also means that the workload cannot be deployed. Workloads that have been deployed to nodes before are not affected.

1. Select **Workloads** from the menu on the left side.
2. Choose the workload you would like to disable.

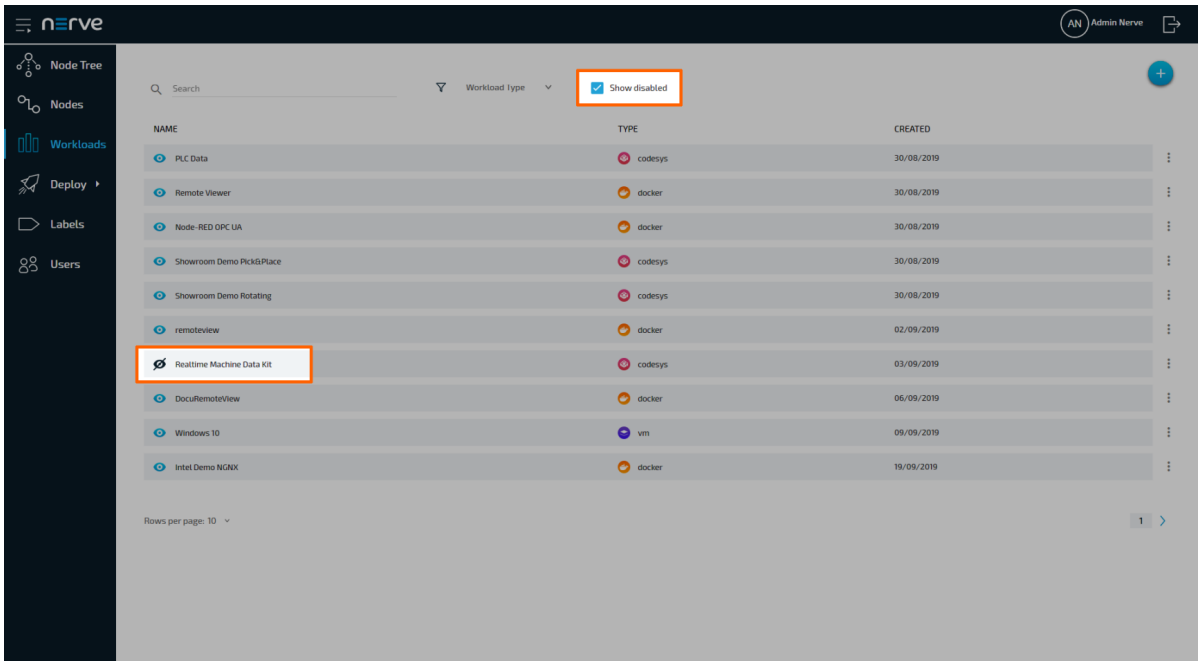


3. Click the ellipsis menu to the right of the workload.
4. Select **DISABLE** from the overlay that appeared.



5. Click **OK** in the new window.

The workload is now disabled and hidden in the list. To show disabled workloads again, tick the checkbox next to **Show Disabled** in the list of workloads. All disabled workloads are marked by an icon resembling a crossed out eye.

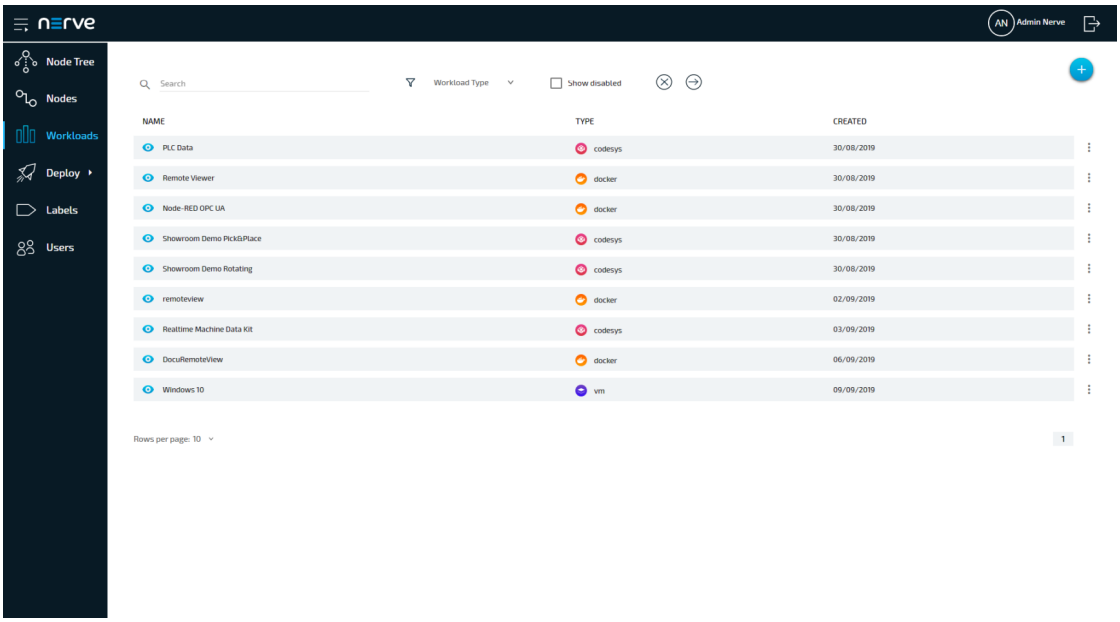


Deleting a Workload

The instructions below cover the deletion of a workload from the repository in the Management System. Note that deleting a workload from the Management System will not automatically remove the workload from nodes. To remove a workload from a node, undeploy the workload in the Node Tree.

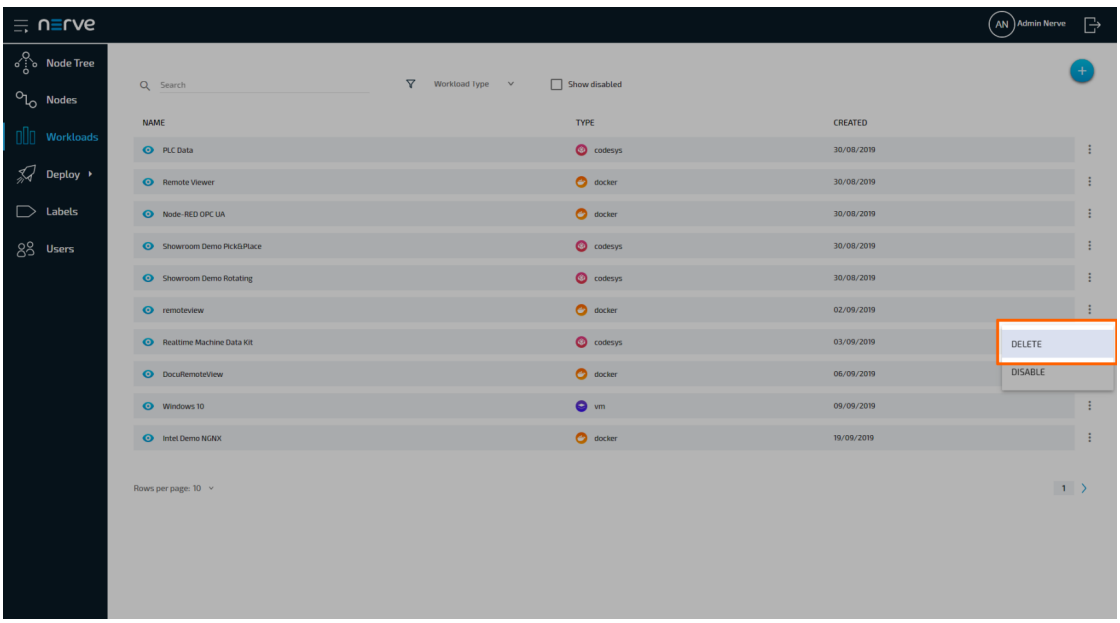
1. Select **Workloads** from the menu on the left side.

2. Choose the workload you would like to delete.



3. Click the ellipsis menu to the right of the workload.

4. Select **DELETE** in the overlay that appeared.



5. Click **OK** in the new window to confirm the deletion.

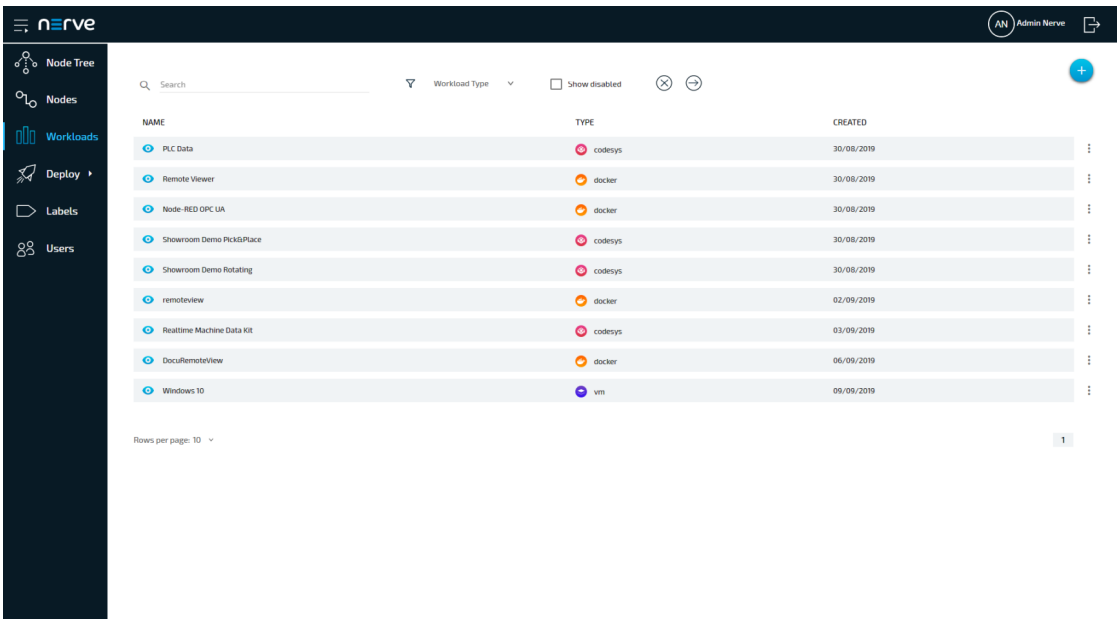
NOTE

Deleting a workload will automatically delete all versions of the workload as well.

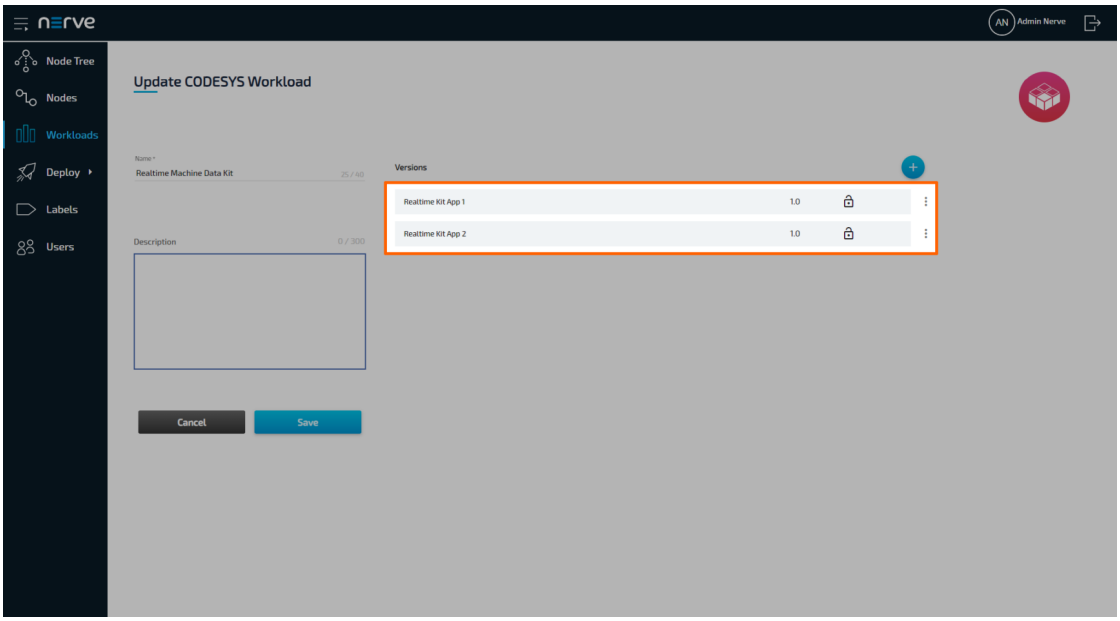
Deleting a Workload Version

Workload versions that are not needed anymore can easily be deleted in the workload details.

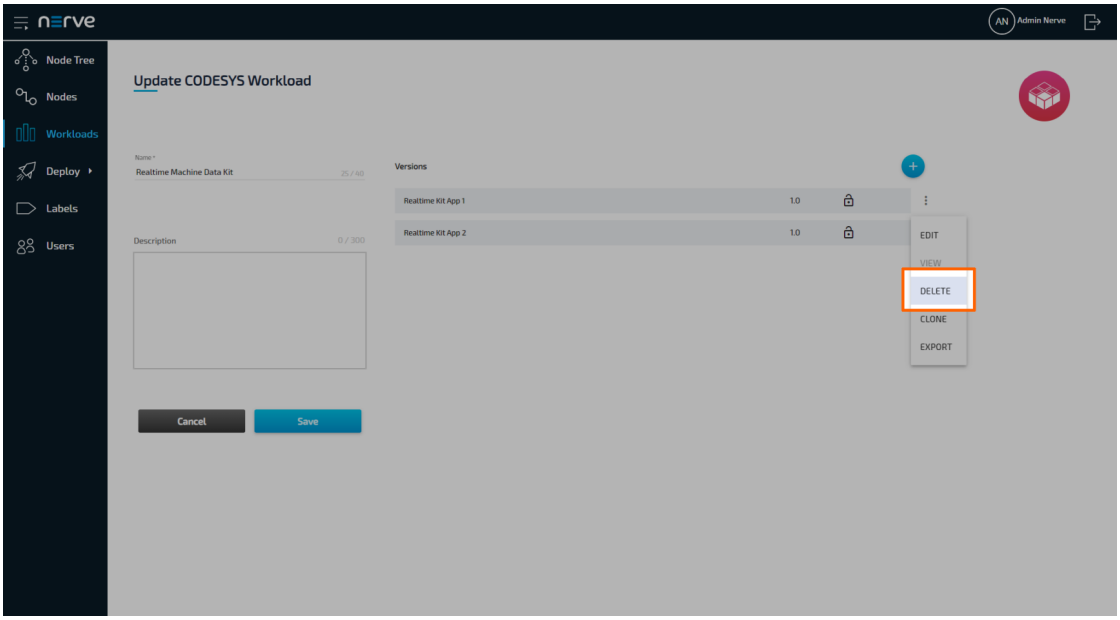
1. Select **Workloads** from the menu on the left side.
2. Select the workload of which you would like to delete a version.



3. Choose the workload version you want to delete.



4. Click the ellipsis menu to the right of the workload version.
5. Select **DELETE** in the overlay that appeared.



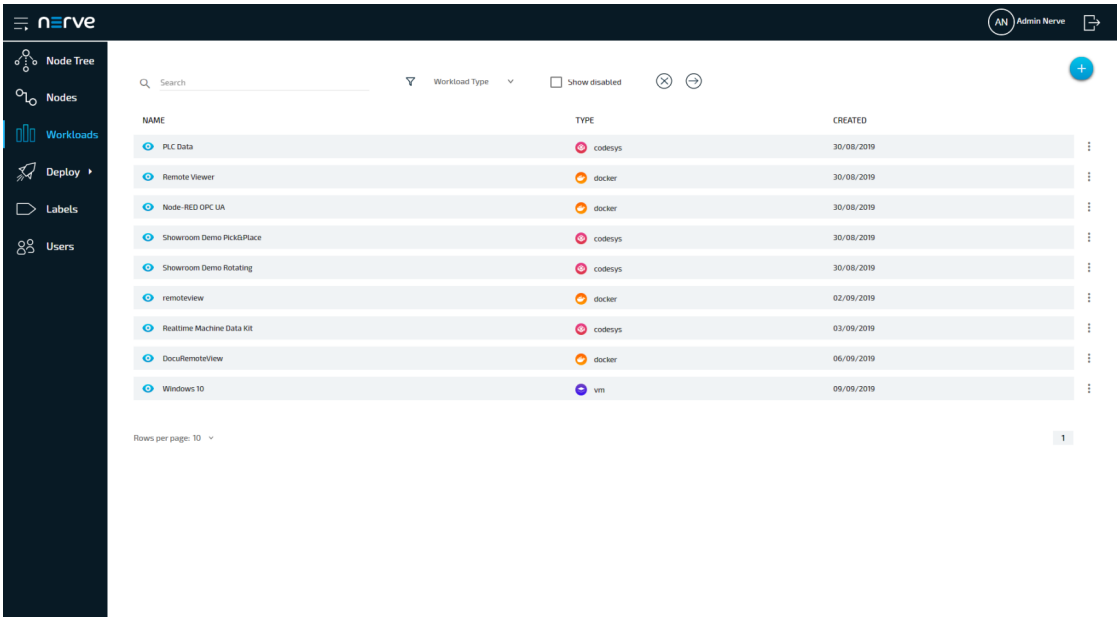
6. Click **OK** in the new window to confirm the deletion.

> **NOTE**
>
> Deleting a workload version is not possible if only one version of the workload exists. If you would like to delete the only version anyway, please delete the entire workload.

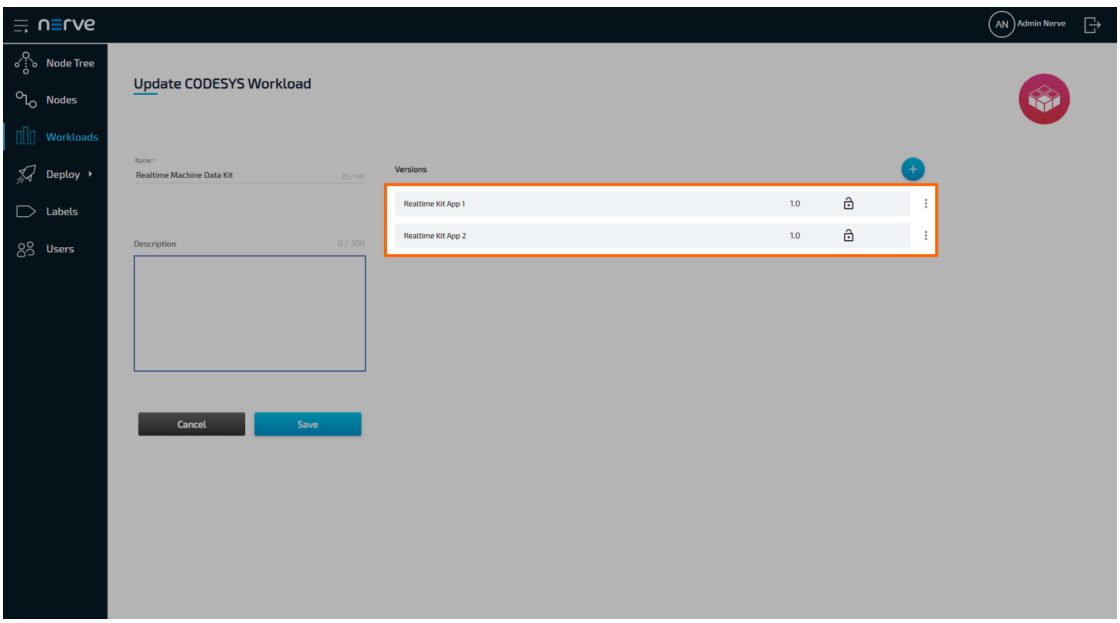
Exporting a Workload

In a future update. it will be possible to manually upload a workload to the Management System or the node. To do that, a workload version must be exported. This is already possible in the current version of the system.

1. Select **Workloads** from the menu on the left side.
2. Select the workload of which you would like to export a version.

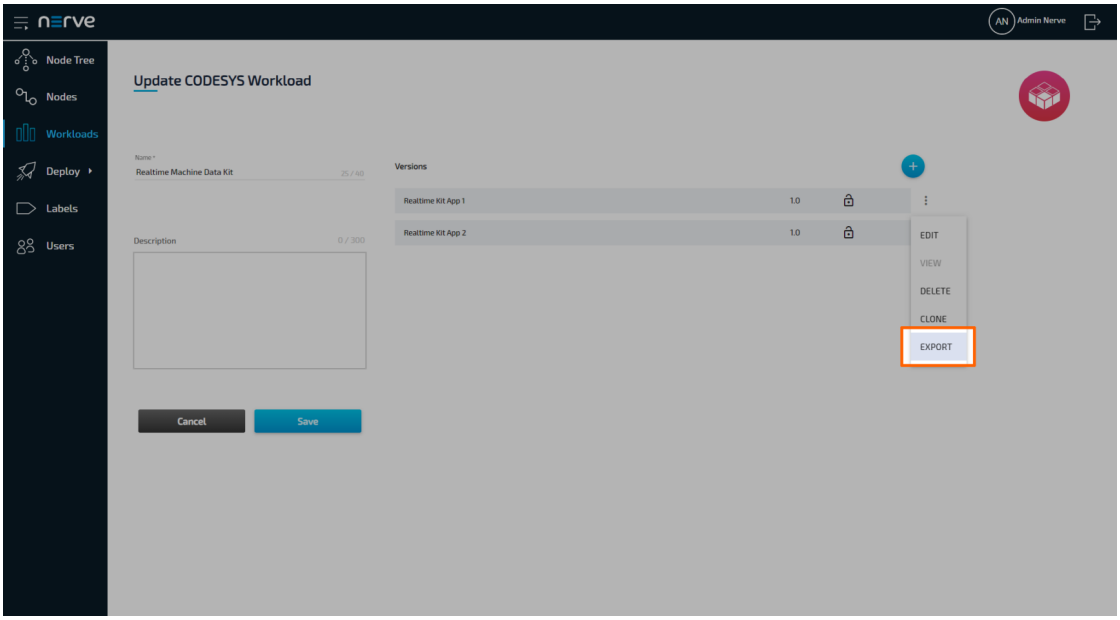


3. Choose the workload version you want to export.



4. Click the ellipsis menu to the right of the workload version.

5. Select **EXPORT** in the overlay that appeared.



The download of the exported workload version is started automatically. The workload version is compressed into a TAR file.

Controlling a Workload

Workloads can be controlled from the node details view in the Node Tree. Please refer to the Node Tree chapter for more information.

Provisioning a CODESYS Workload

Before a CODESYS workload can be provisioned, a CODESYS application has to be loaded into the CODESYS runtime first. Refer to the introduction to working with CODESYS and the Nerve Device first before continuing.

Once a CODESYS application has been loaded into the CODESYS runtime, the following steps have to be taken before the workload can be provisioned:

1. Creating the ZIP file of the CODESYS application
2. Transferring the ZIP file to a local workstation

In this version, Nerve Blue does not provide a GUI based method for creating a ZIP file of the CODESYS application. Therefore, this chapter focuses on the manual process. Two tools are required for the instructions below, assuming Windows is used on the workstation:

- an SSH client like PuTTY
- a file transfer client like WinSCP

Also the workstation needs to be connected to the management port of the Nerve Device and the IP address of the network adapter of the workstation needs to be configured in the correct range. This information is device specific. Consult the device guide for information on your Nerve Device.

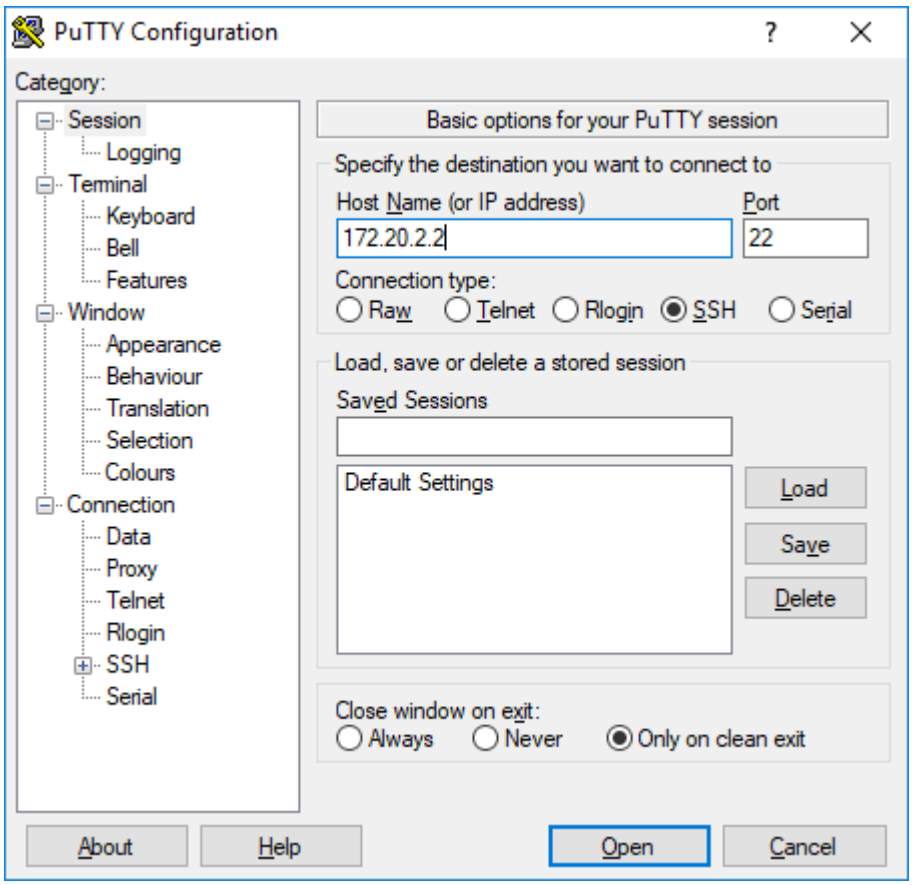
Creating the ZIP File on the Nerve Device

NOTE

The instructions below are hardware specific. The MFN 100 is used as an example in the screenshots. See the device guide for specific information on your Nerve Device.

First, the CODESYS project needs to be zipped on the Nerve Device before it can be copied from the CODESYS runtime.

1. Open an SSH client like PuTTY.
2. Enter the IP address of the CODESYS runtime under **Host Name (or IP address)** to log in to the CODESYS runtime on the Nerve Device.



3. Click **Open**.
4. In the new window, enter the admin password for the CODESYS runtime.
5. Enter the following commands:

   sudo su

   systemctl stop nerve-codesys

   cd/var/lib/codesys

zip -r /tmp/<userproject>.zip CmpApp.cfg PlcLogic/

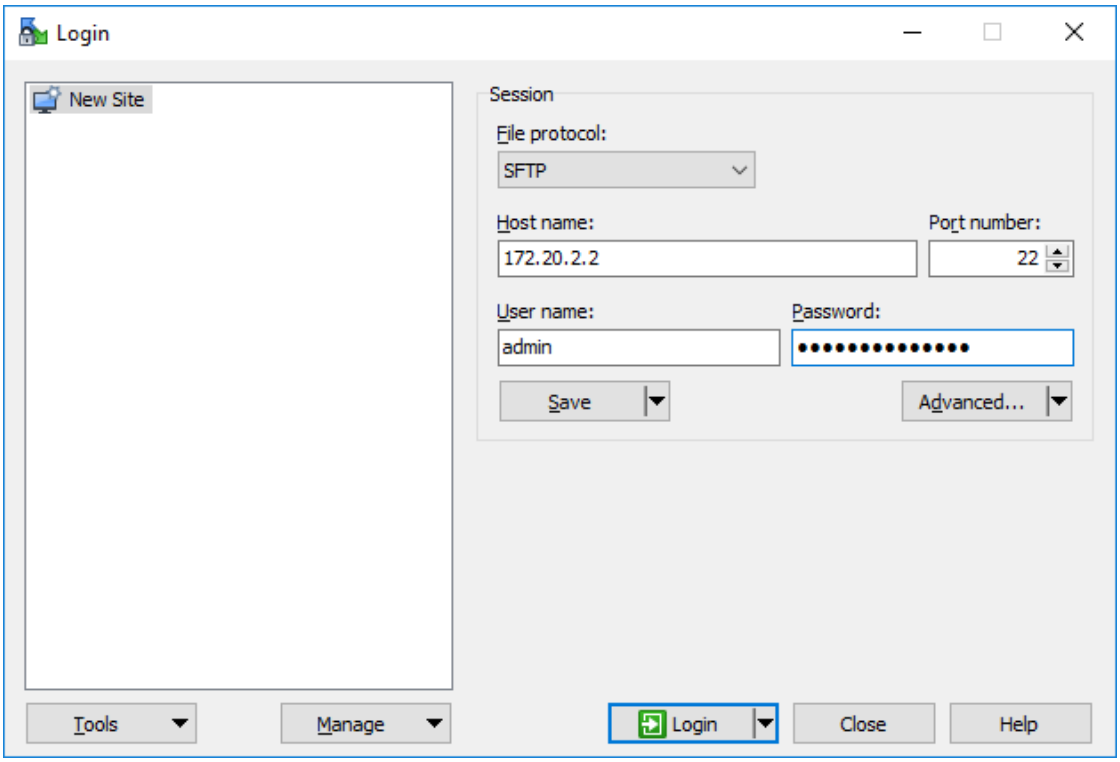
> **NOTE**
>
> You can freely name the ZIP file here by replacing <userproject> with a name of your choice.

The ZIP file has now been created and is available on the Nerve Device.

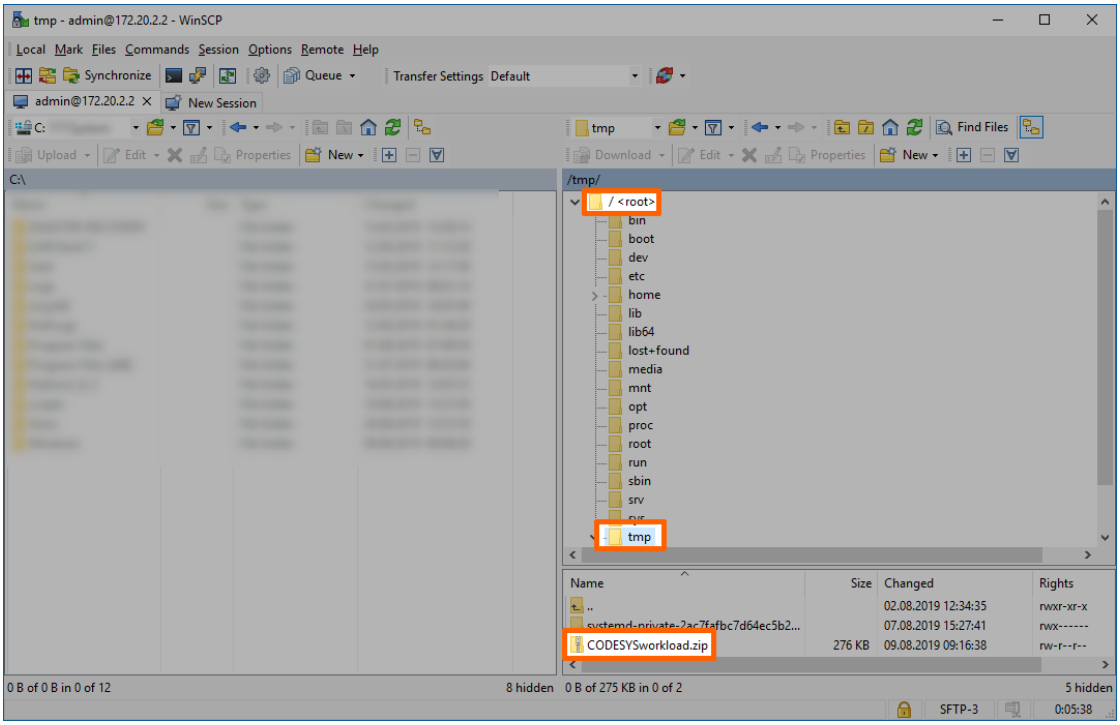
Transferring the ZIP File to a Local Workstation

After zipping the CODESYS project on the Nerve Device, you need to transfer it with a file transfer client to your workstation.

1. Open a file transfer client like WinSCP.
2. Enter the IP address of the CODESYS runtime under **Host Name**.
3. Enter the credentials for the CODESYS runtime below under **User name** and **Password**.



4. Navigate to the **/tmp/** directory on the right side of the WinSCP window. You will find it in the **root** directory.



5. Drag and drop the ZIP file to your workstation.

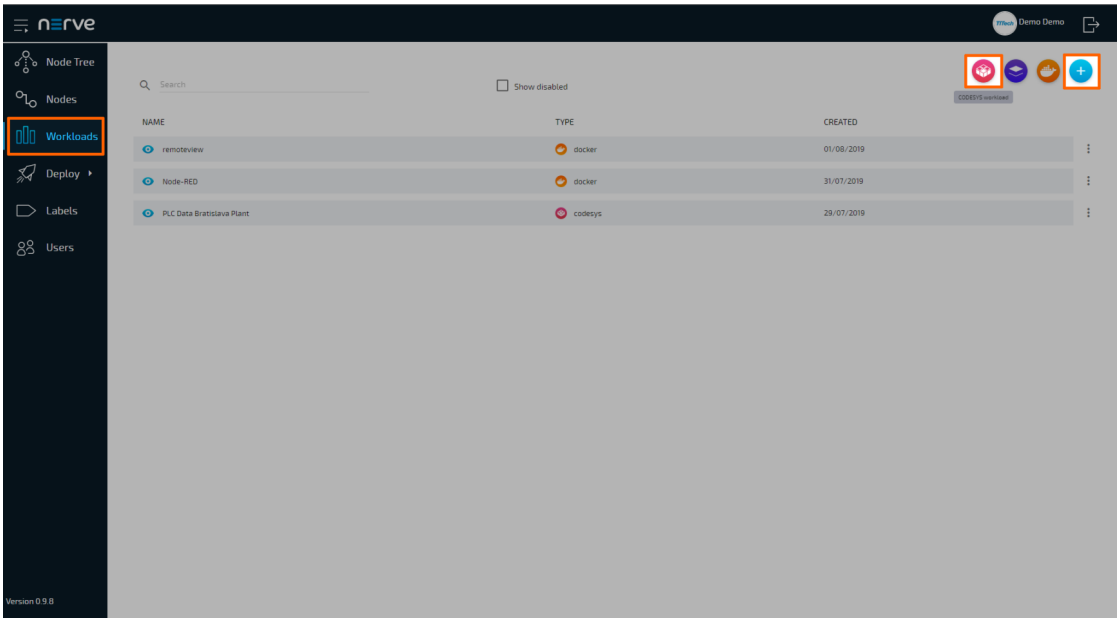
Since the CODESYS project ZIP file is on your local workstation, you can now provision a CODESYS workload in the Management System.

Provisioning a CODESYS Workload

The following instructions cover the basic requirements for provisioning a CODESYS workload. Optional settings will be left out. Extended options are addressed in the last section of this chapter.
There are two further types of workloads that can be provisioned: Virtual Machine workloads and Docker workloads. The process for each workload is highlighted in its respective chapter.

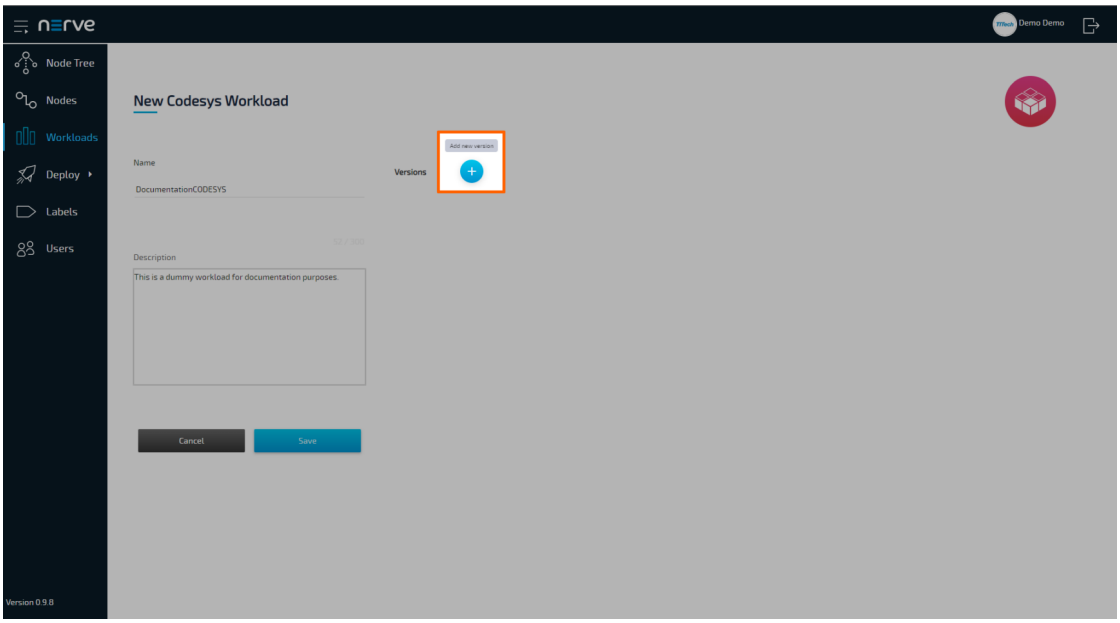
1. Log in to the Management System.
2. Select **Workloads** in the left-hand menu.
3. Select **+** in the upper-right corner.

4. Select the CODESYS symbol (**CODESYS workload**) on the left of the three symbols that expanded.



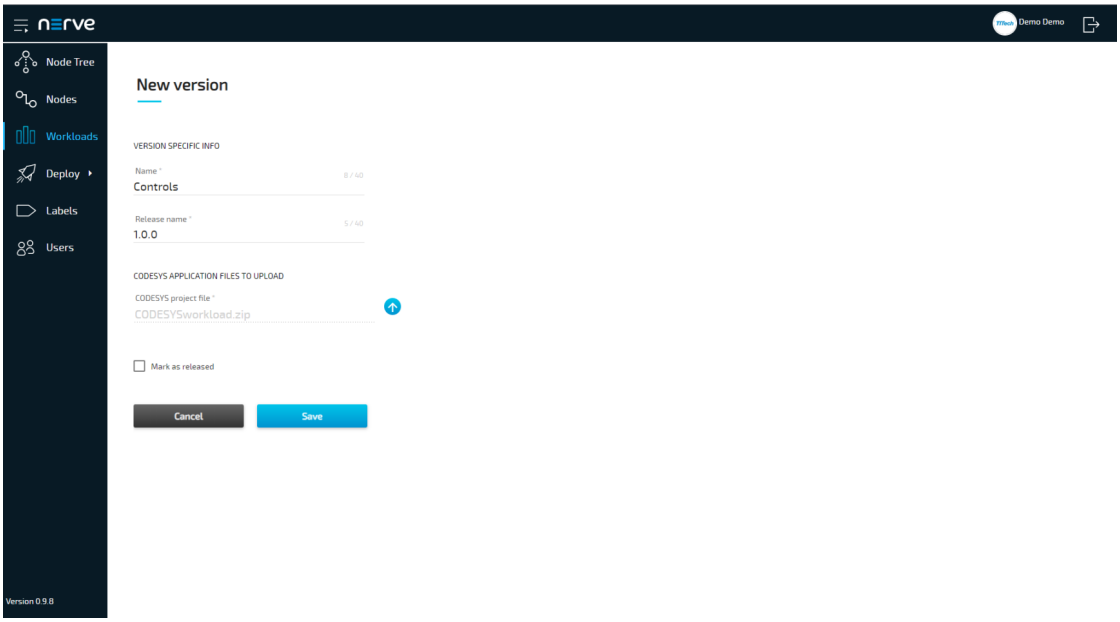
5. In the new window, enter a name for your workload.

6. Select **+** next to **Versions** to add a new version of the workload.



7. In the new window, enter the following information:

Item	Description
Name	Enter a **Name** for the version of this workload.
Release name	Enter a **Release name** for the version of this workload.
CODESYS project file	Click the **upward arrow** symbol to add the CODESYS application ZIP file. This is the ZIP file that you have created before.



8. Click **Save**.

The workload has now been provisioned and is ready to be deployed in the **Deploy** menu.

Settings for CODESYS Workloads

In the instructions above, optional settings have been left out. Below is an overview of all options with an explanation to each option.

Setting	Description
VERSION SPECIFIC INFO	**Name** A name for your workload version. Choose a precise name to make the workload version unambiguous. **Release name** A release name for your workload version. This could be a version number. Example: 1.0.1
CODESYS APPLICATION FILES TO UPLOAD	Here you have to upload the **CODESYS project file**. This is a ZIP file that has to be generated from a CODESYS project running in the CODESYS runtime. Upload it here by clicking the **upward arrow** symbol to open your file browser.
SELECTOR	**Labels** If you have defined labels and assigned them to nodes, you can add them as selectors to the workload. When deploying a workload, the list of nodes will be filtered automatically to the specified label.
Mark as released	Tick this checkbox if you want to mark this workload as released. Once marked as released, the workload cannot be edited anymore.

Provisioning a Virtual Machine Workload

Before a Virtual Machine workload can be provisioned, it is required to set up the virtual machine on the node. In more detail, what needs to be done is:

1. Creating a virtual machine on the node
2. Installing an operating system on the virtual machine
3. Obtaining the virtual machine IMG and XML files
4. Provisioning the Virtual Machine workload in the Management System

In this version, Nerve Blue does not provide a GUI based method for installing an OS on a virtual machine and obtaining the virtual machine IMG and XML configuration files. Therefore, this chapter focuses on the manual process. Three tools are required for the instructions below, assuming Windows is used on the workstation:

- an X Server application like Xming
- an SSH client like PuTTY
- a file transfer client like WinSCP

Also the workstation needs to be connected to the management port of the Nerve Device and the IP address of the network adapter of the workstation needs to be configured in the correct range. This information is device specific. Consult the device guide for information on your Nerve Device.

The instructions below are split up into multiple parts to make them easier to follow. The subsections of the instructions are connected and every subheading is a requirement for the next paragraph.

NOTE

The virtual machine generated in this chapter is a fresh installation and will be generated on the Nerve Device directly.

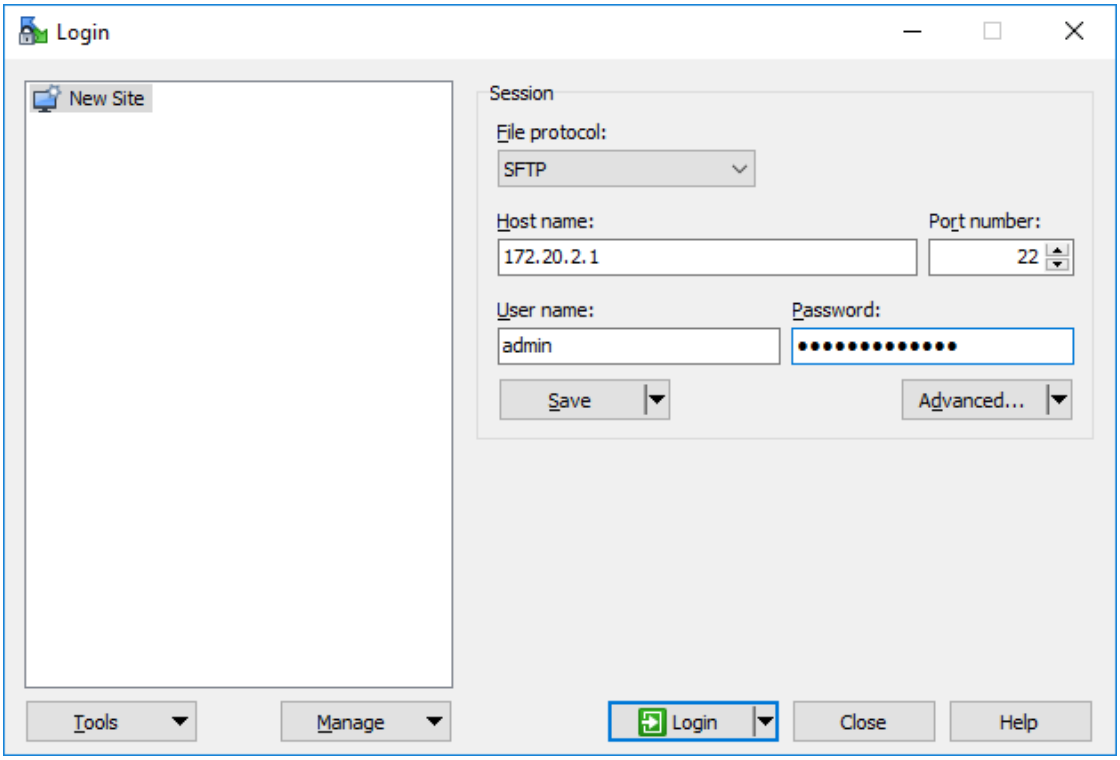
Copying the ISO File to the Nerve Device

First, it is necessary to copy the ISO file of the OS to the Nerve Device for the installation on the virtual machine.

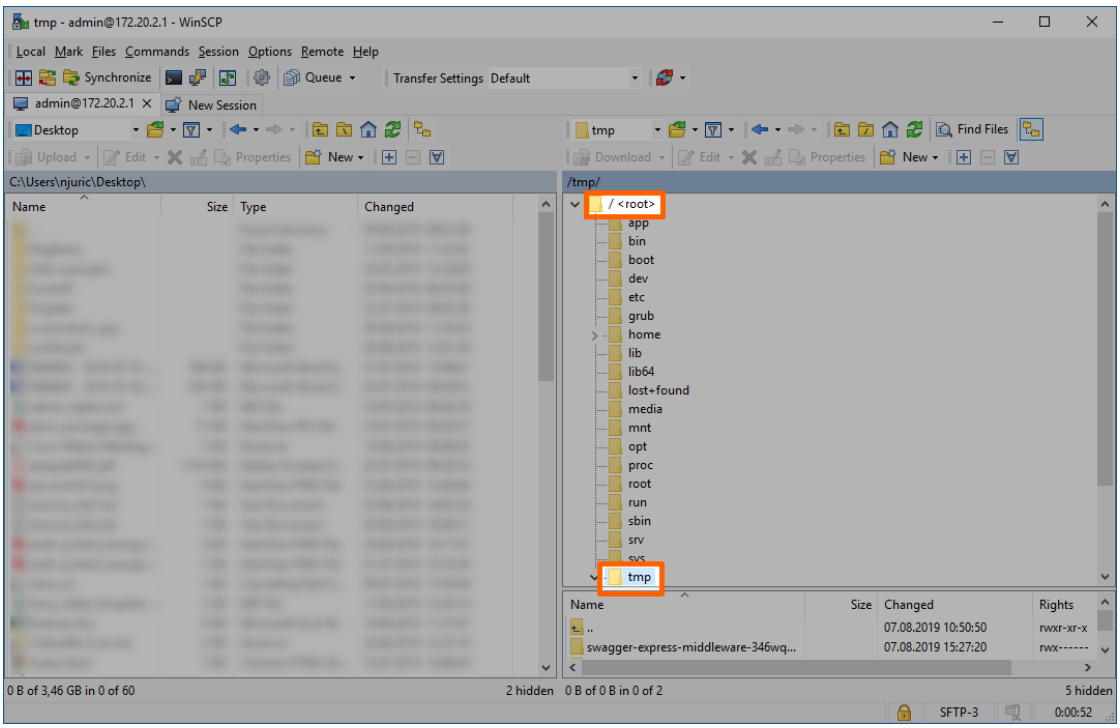
NOTE

The instructions below are hardware specific. The MFN 100 is used as an example in the screenshots. See the device guide for specific information on your Nerve Device.

1. Open a file transfer client like WinSCP.
2. Enter the IP address for host access of your Nerve Device under **Host Name**.
3. Enter the credentials for host access to the Nerve Device below under **User name** and **Password**.



4. Navigate to the **/tmp/** directory on the right side of the WinSCP window. It is located in the **root** directory.



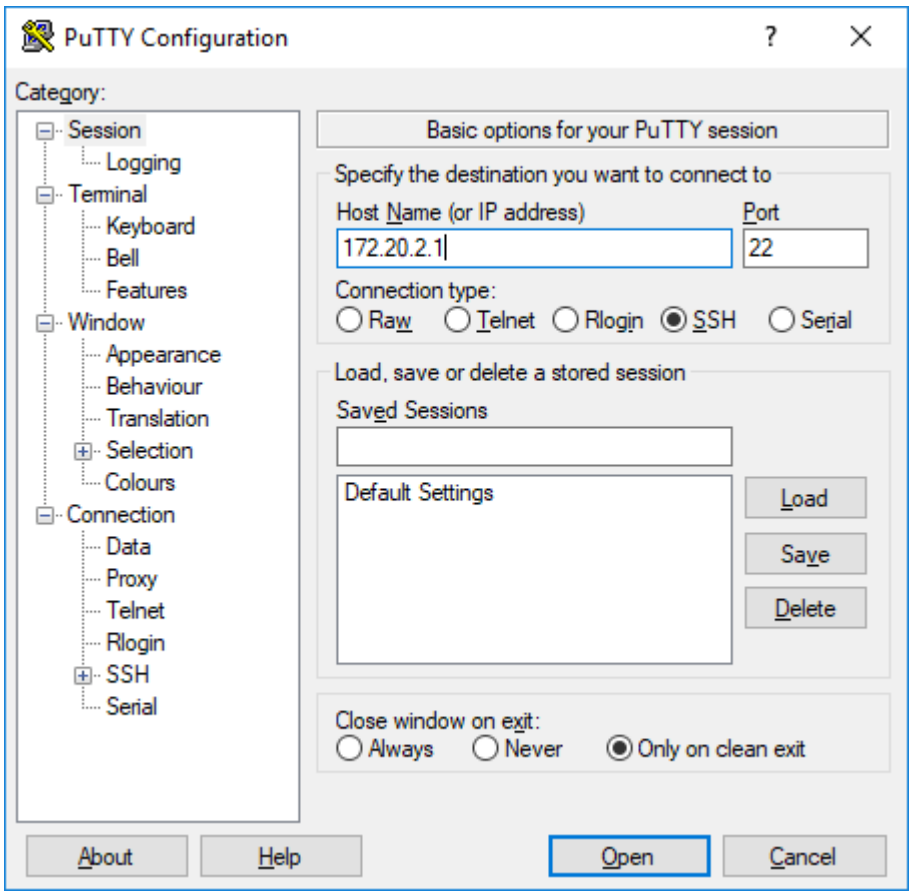
5. Copy the ISO file of the OS that is to be installed on the virtual machine to the**tmp** folder on the Nerve Device.

Creating a Virtual Machine on a Node

Using the Virtual Machine Manager is recommended to create a virtual machine and install the OS from the ISO file. Note that the virtual machine in this chapter is a fresh installation and will be generated on the Nerve Device directly.

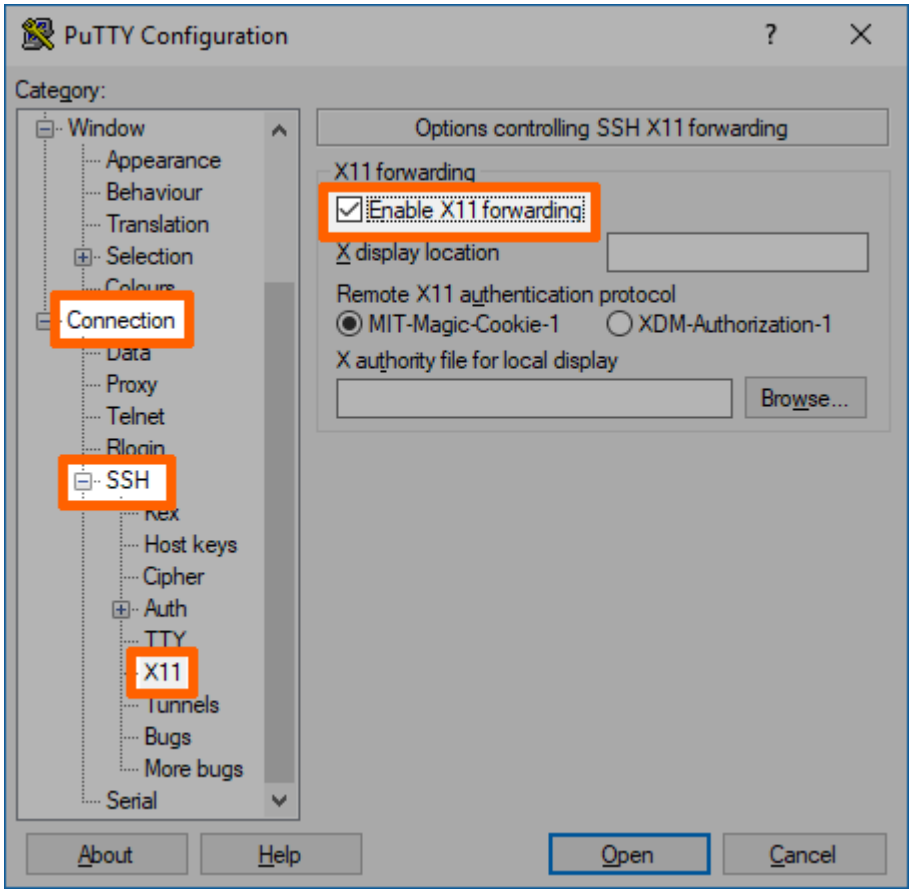
1. Run Xming or an alternative.
2. Open an SSH client like PuTTY.

3. Enter the IP address for host access of your Nerve Device under **Host Name (or IP address)** to log in to the host of the Nerve Device.



4. Expand **Connection > SSH > X11** on the left side.

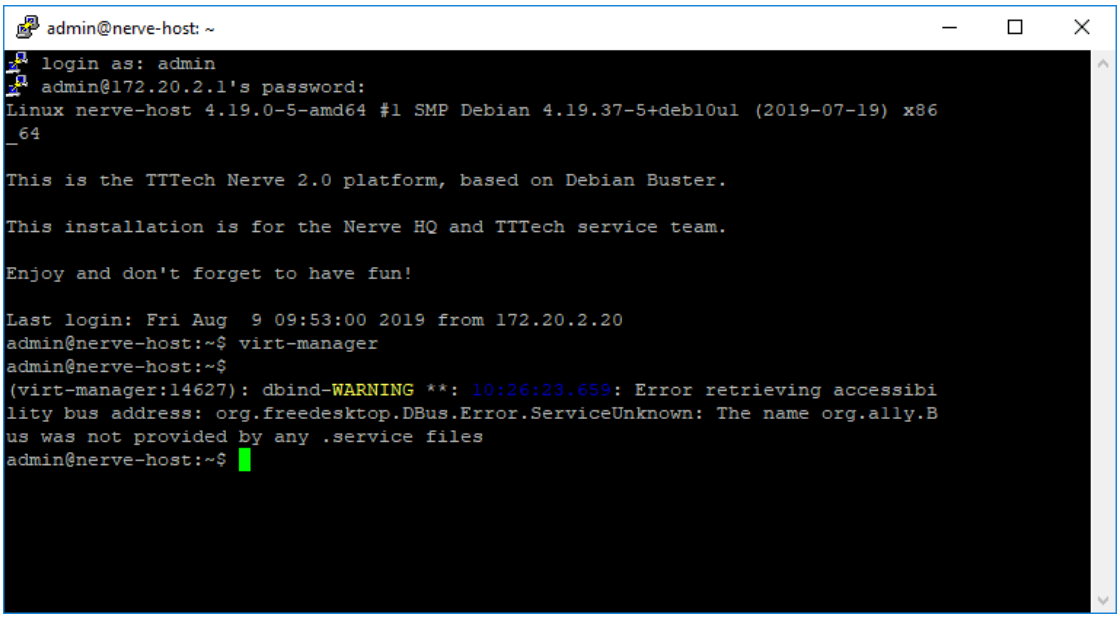
5. Tick the checkbox next to **Enable X11 forwarding**.



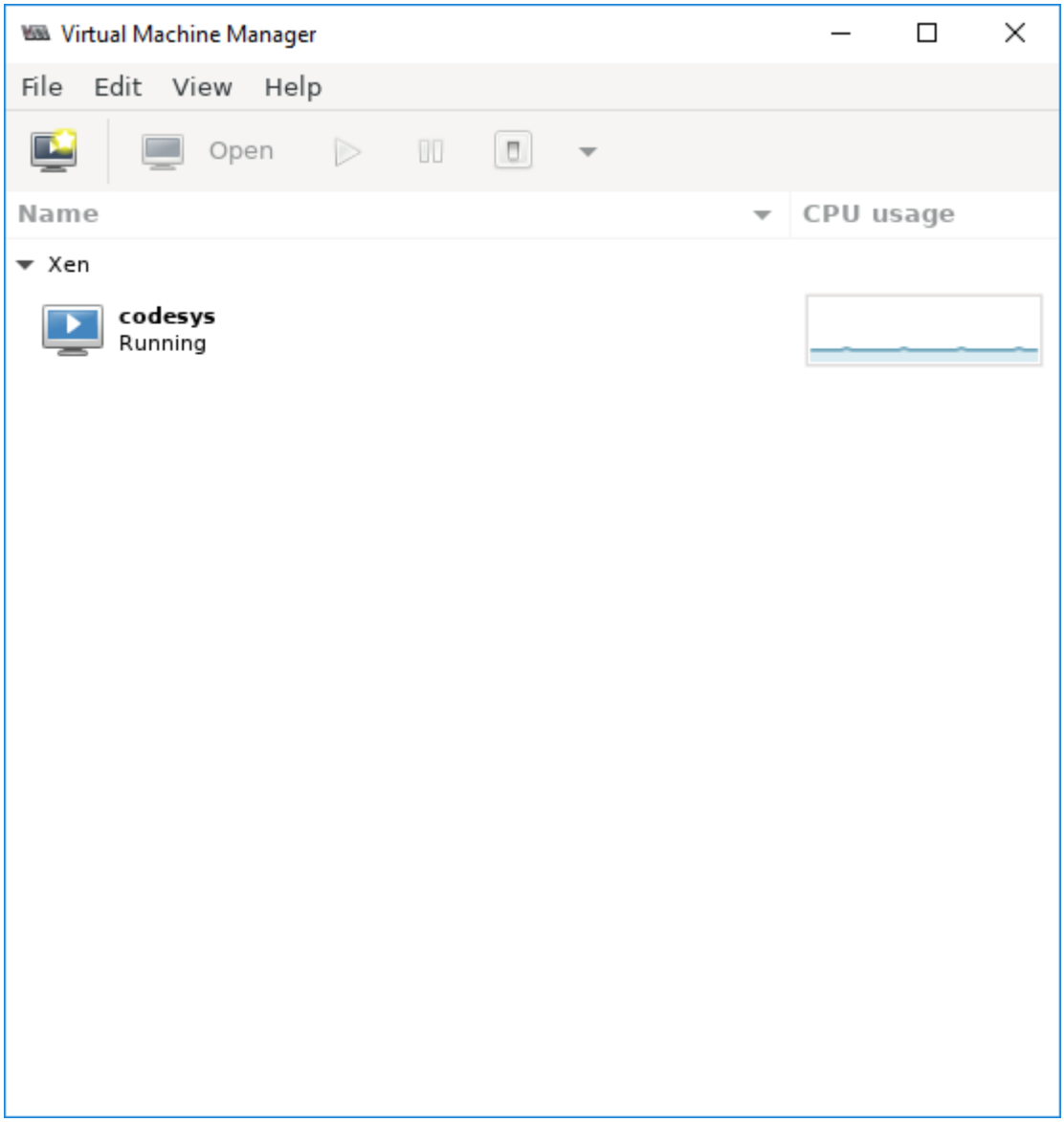
6. Click **Open**.

7. Log in with the credentials for host access of the Nerve Device.

8. Enter virt-manager.



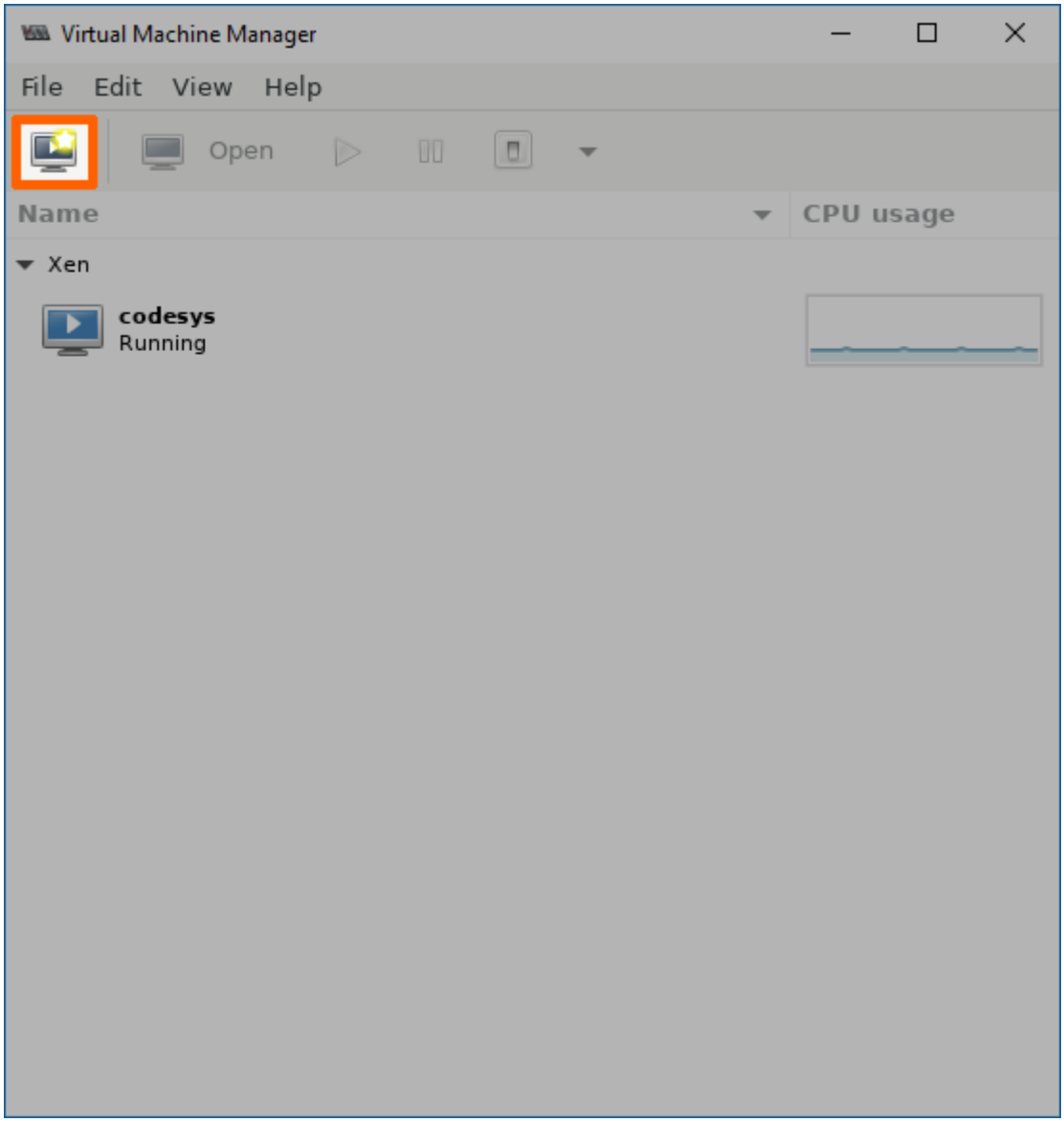
The interface of the Virtual Machine Manager will open.



Inserting the Installation File (ISO)

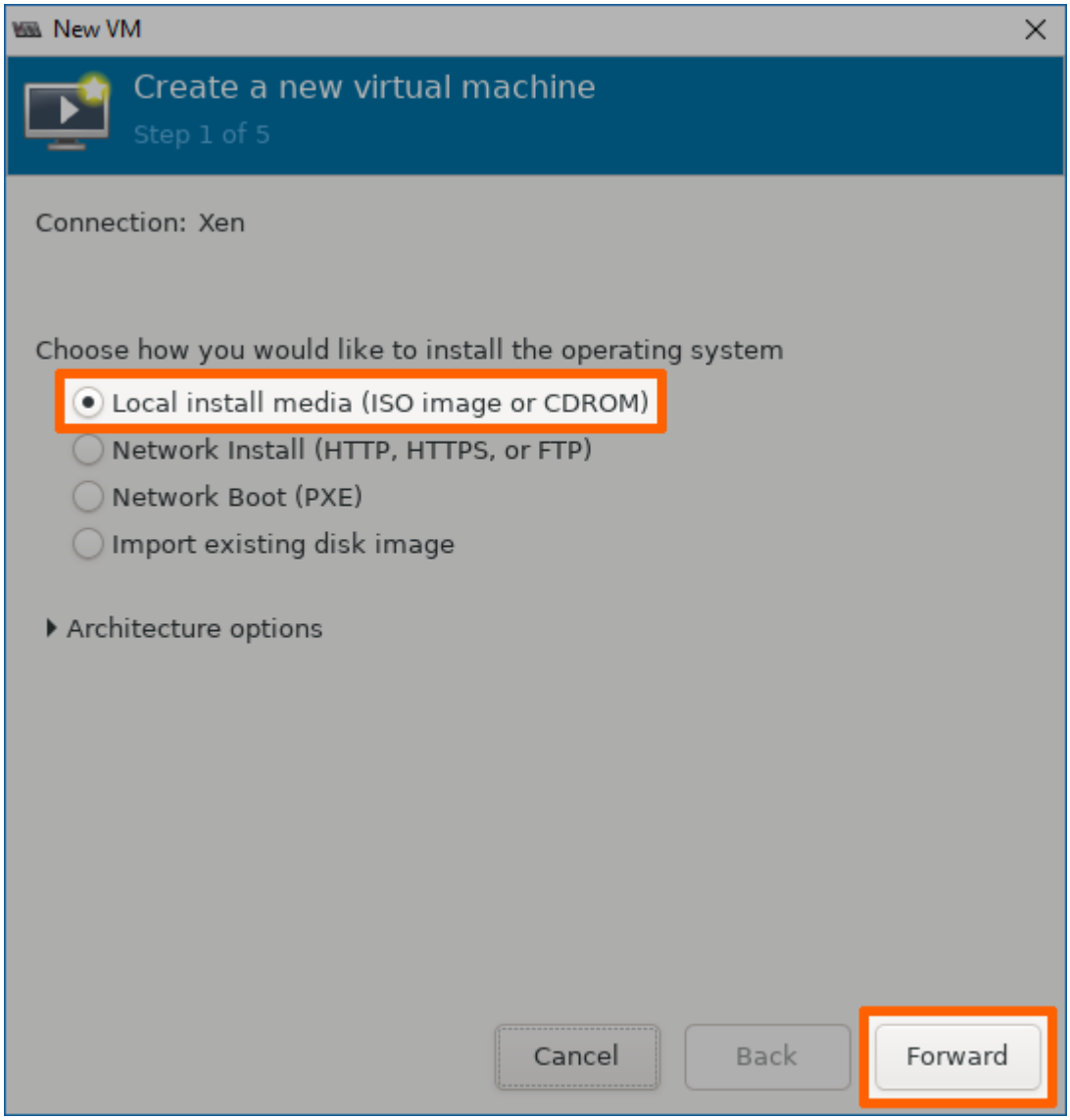
The creation of the virtual machine can now be initiated with the installation of the OS following right after. Note that the Virtual Machine Manager requires the virtual insertion of the ISO file in the beginning while resources for the virtual machine are defined later.

1. Select **File > New Virtual Machine** or click the symbol.



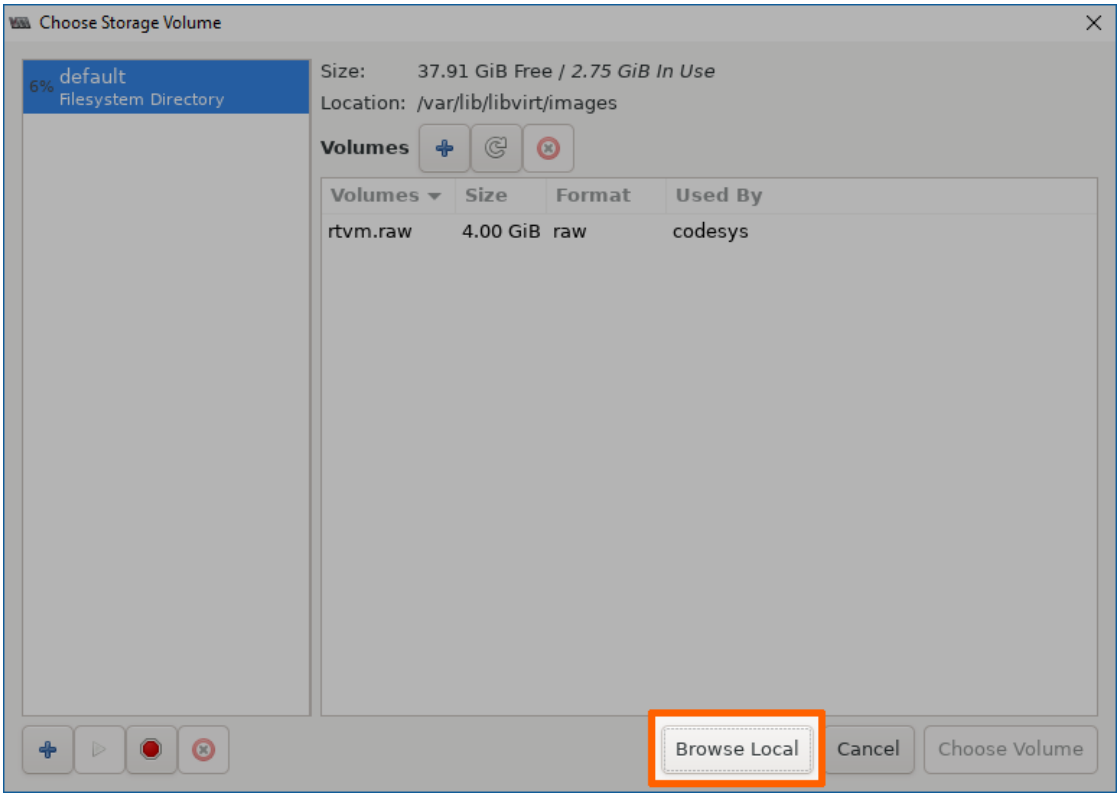
2. Select **Local install media (ISO image or CDROM)**.

3. Click **Forward**.



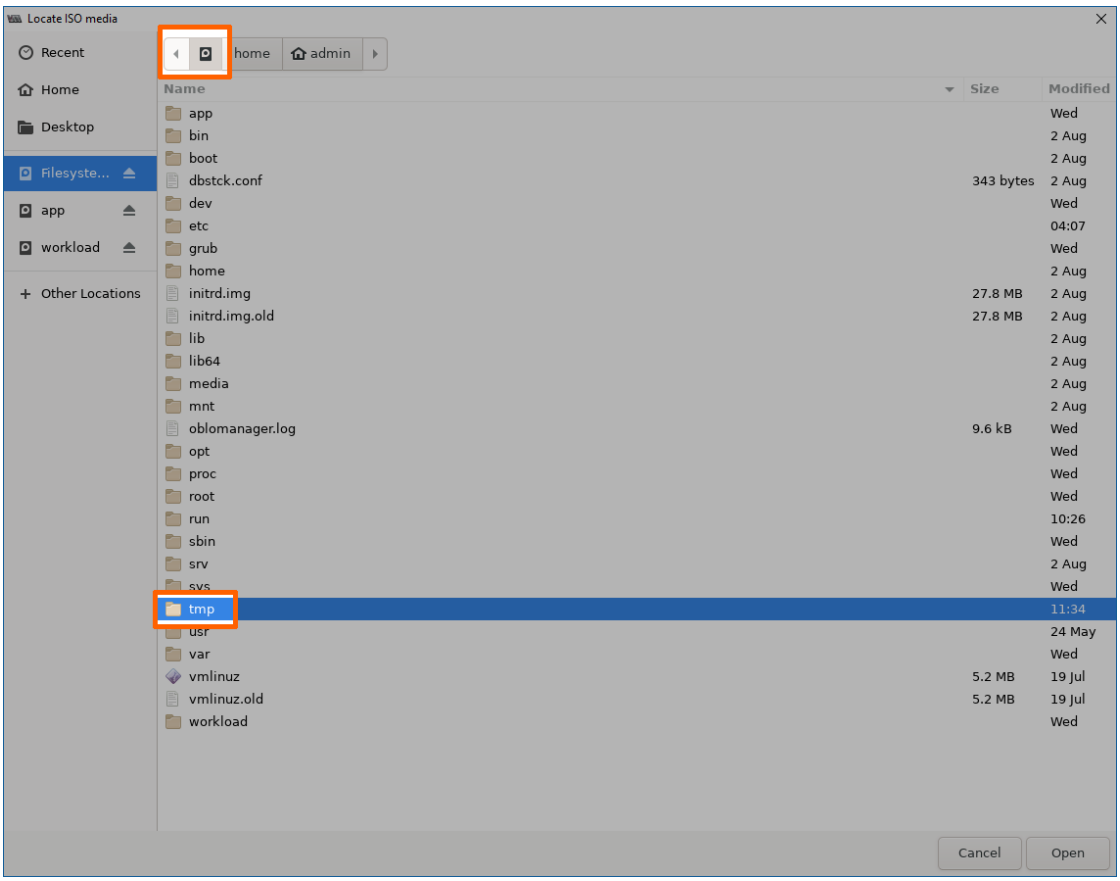
4. Click **Browse...** in the next window.

5. In the new window select **Browse Local**.



6. Navigate to the **tmp** folder by clicking the left arrow next to **admin**. It is located in the root directory.

7. Double-click the **tmp** folder.



Select the ISO file of the OS that you have copied before.

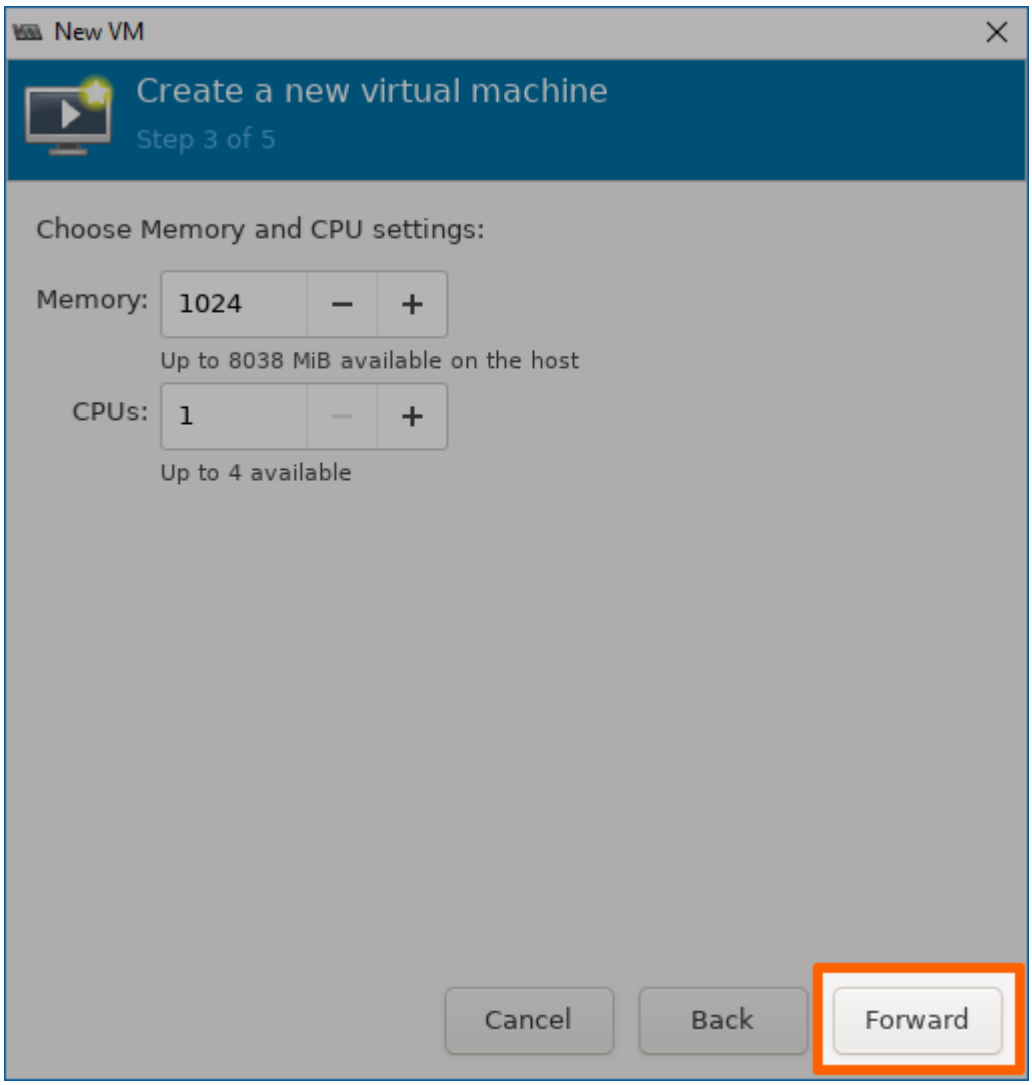
8.
9. Click **Open**.

Defining Resources for the Virtual Machine

Next, the amount of memory, the number of CPUs and the size of the logical volume assigned to the virtual machine need to be defined.

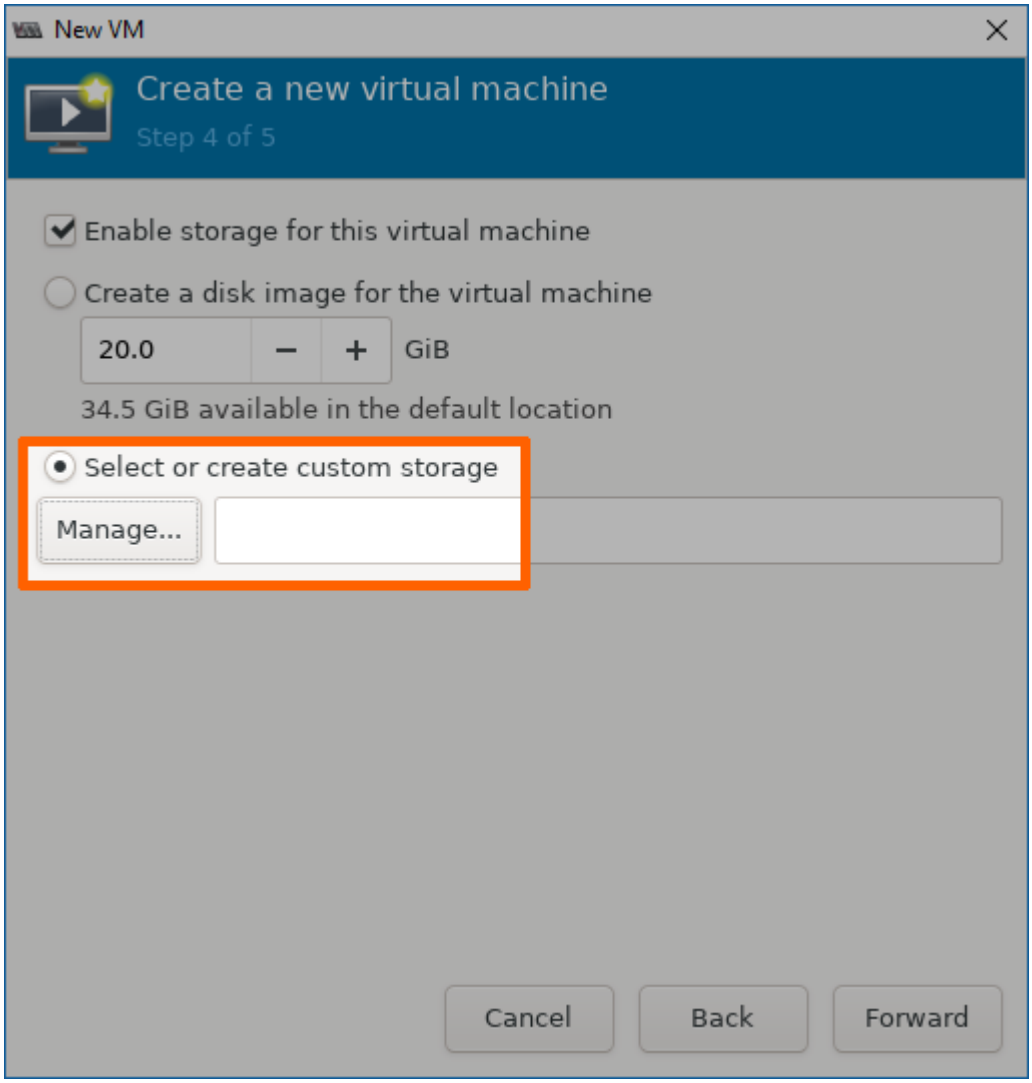
1. Click **Forward**.
2. Define how much memory and how many CPUs to assign to this virtual machine.

3. Select **Forward**.

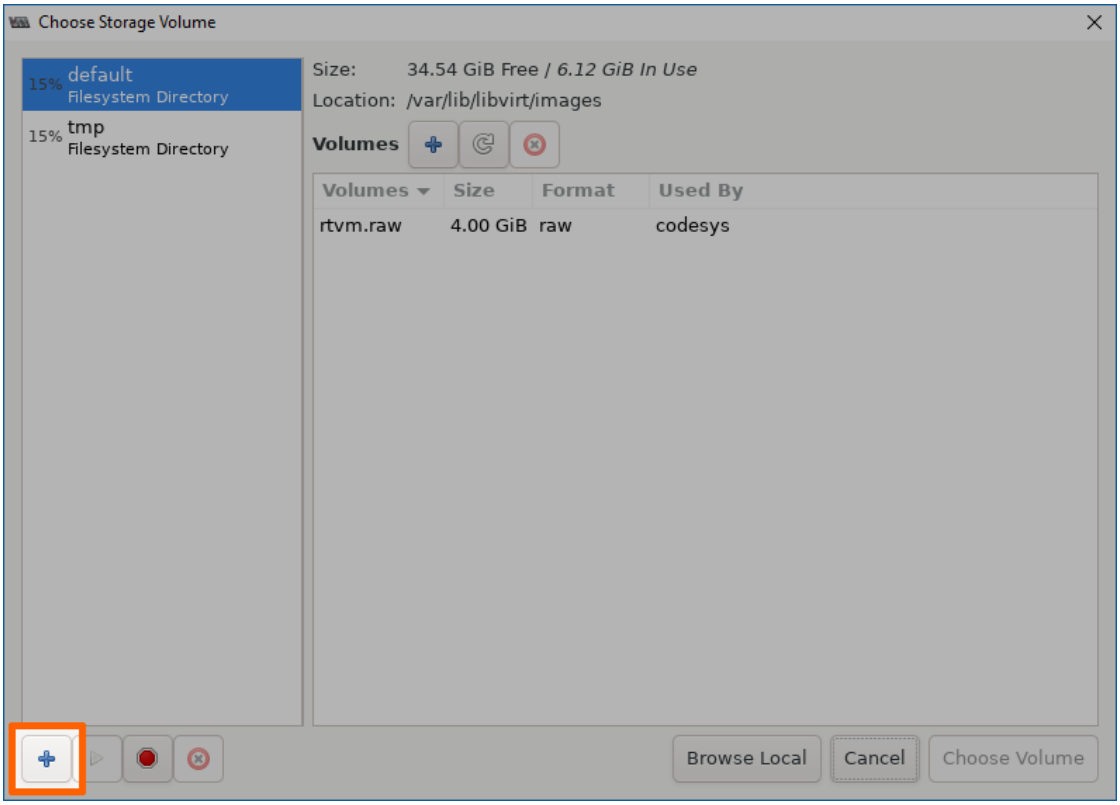


4. Click **Select or create custom storage**.

5. Select **Manage...**.



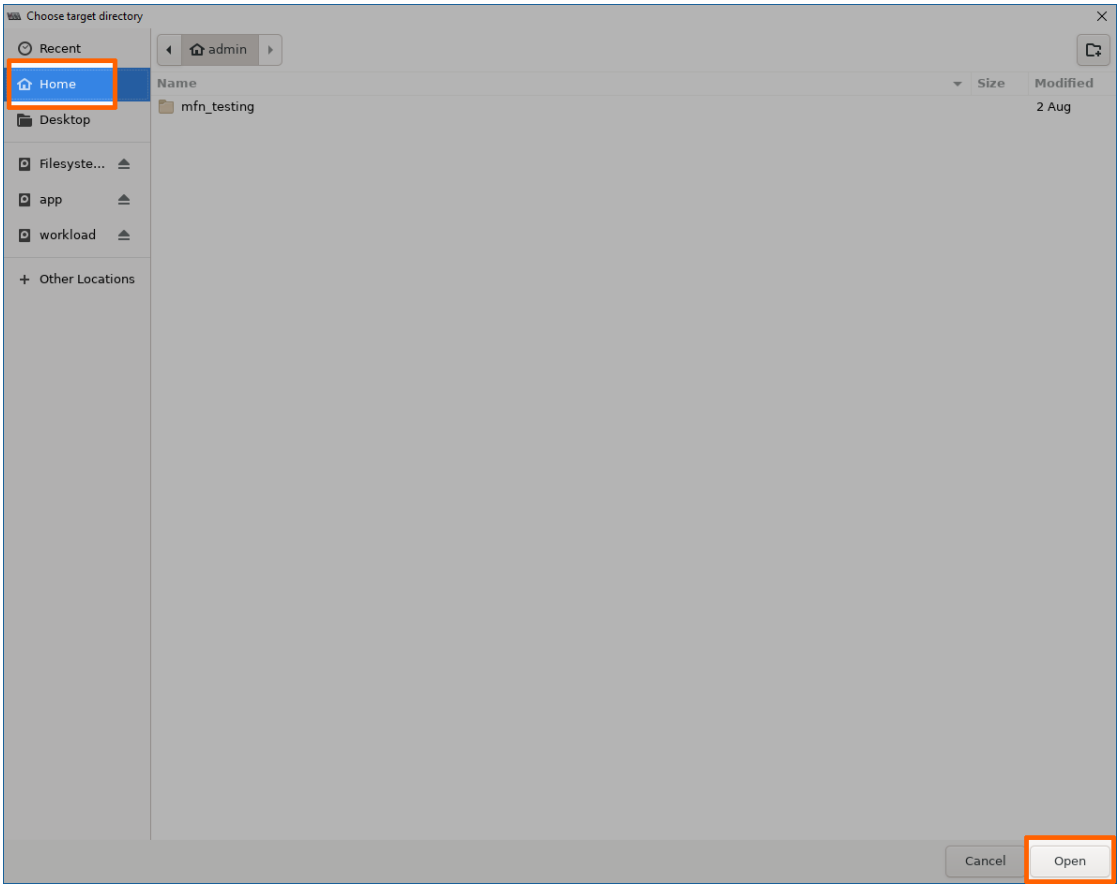
6. Click the **+** icon in the lower-left corner to add a new logical volume group (**Add Pool**).



7. Enter a name for the storage pool type.

8. Click **Forward**.
9. Click **Browse**.
10. Select **Home** on the left side.

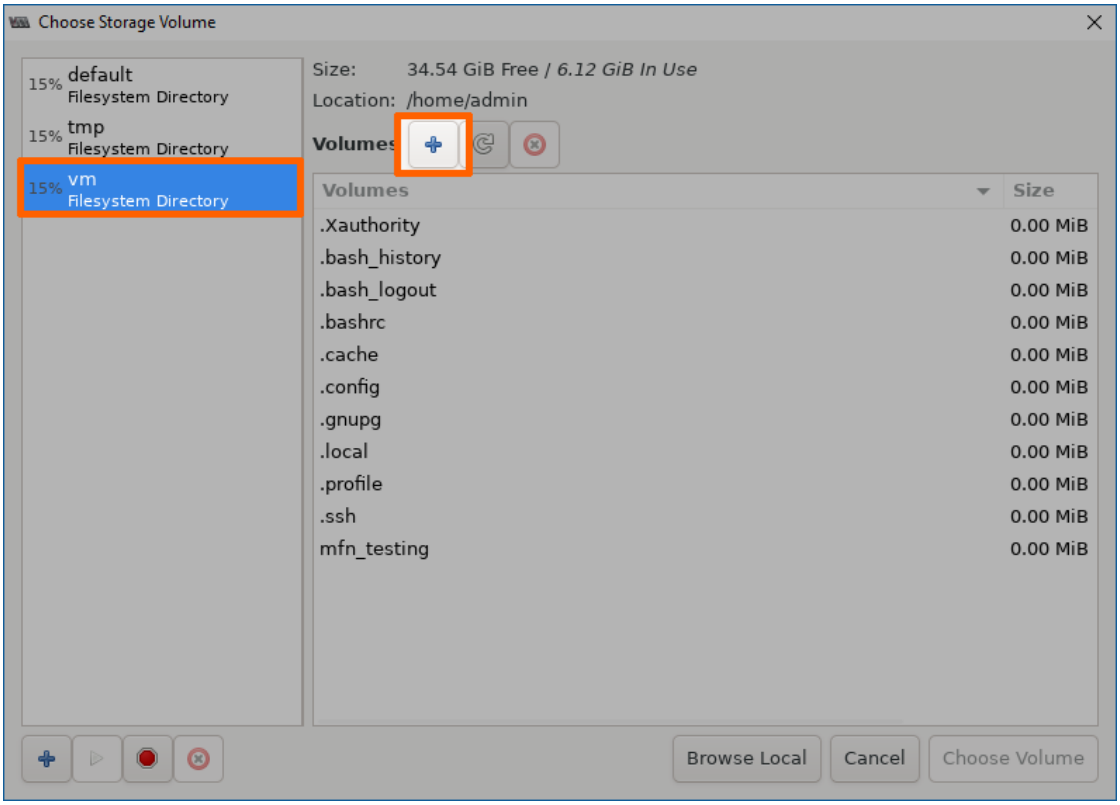
11. Click **Open** in the lower-right corner.



12. Select **Finish**.

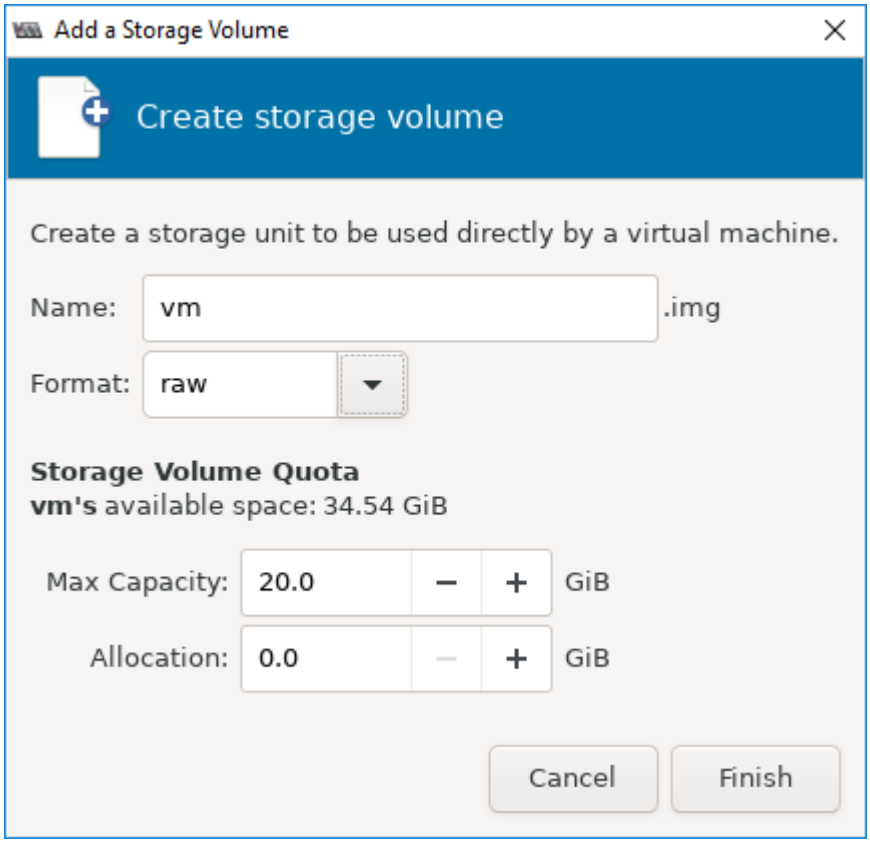
13. Select the new pool you have just created on the left side.

14. Click the **+** symbol next to **Volumes**.



15. Enter the following information in the new window:

Item	Description
Name	Enter a name for the storage volume used by the virtual machine.
Format	Select **raw** from the drop-down menu.
Max Capacity	Choose the storage capacity of the storage volume.



NOTE

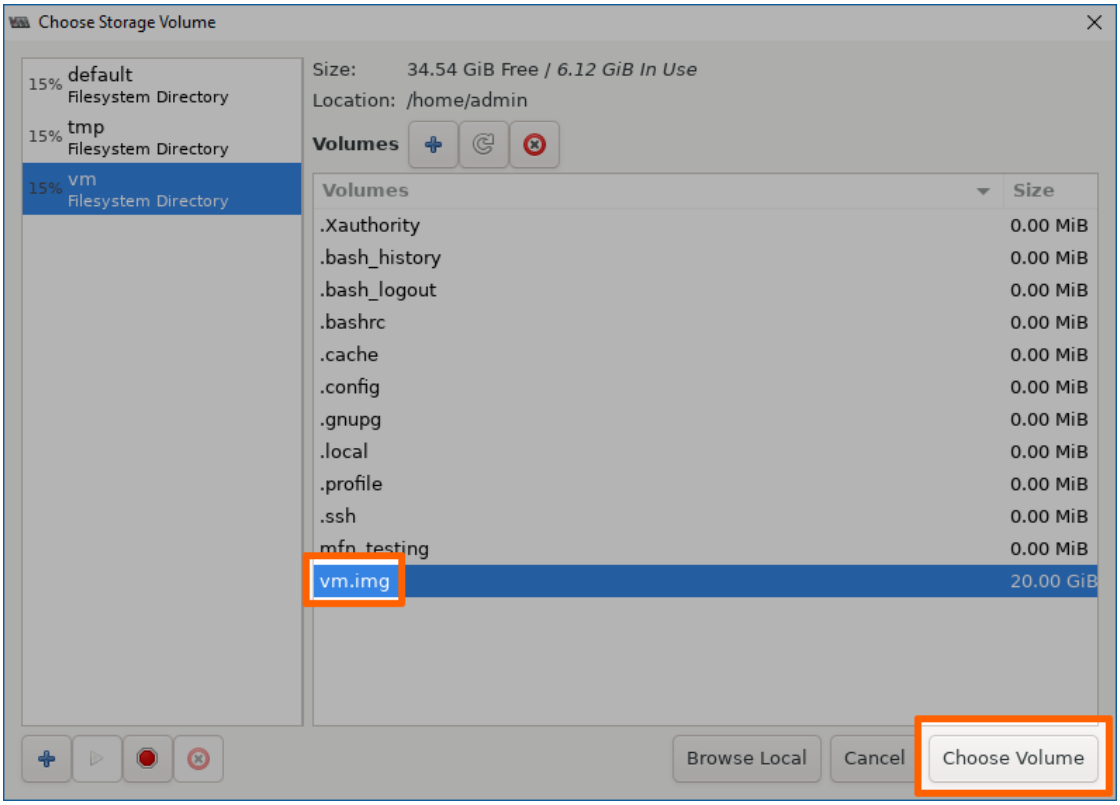
The value in the screenshot is a placeholder. Please adjust the capacity according to the virtual machine you are generating.

16. Click **Finish**.

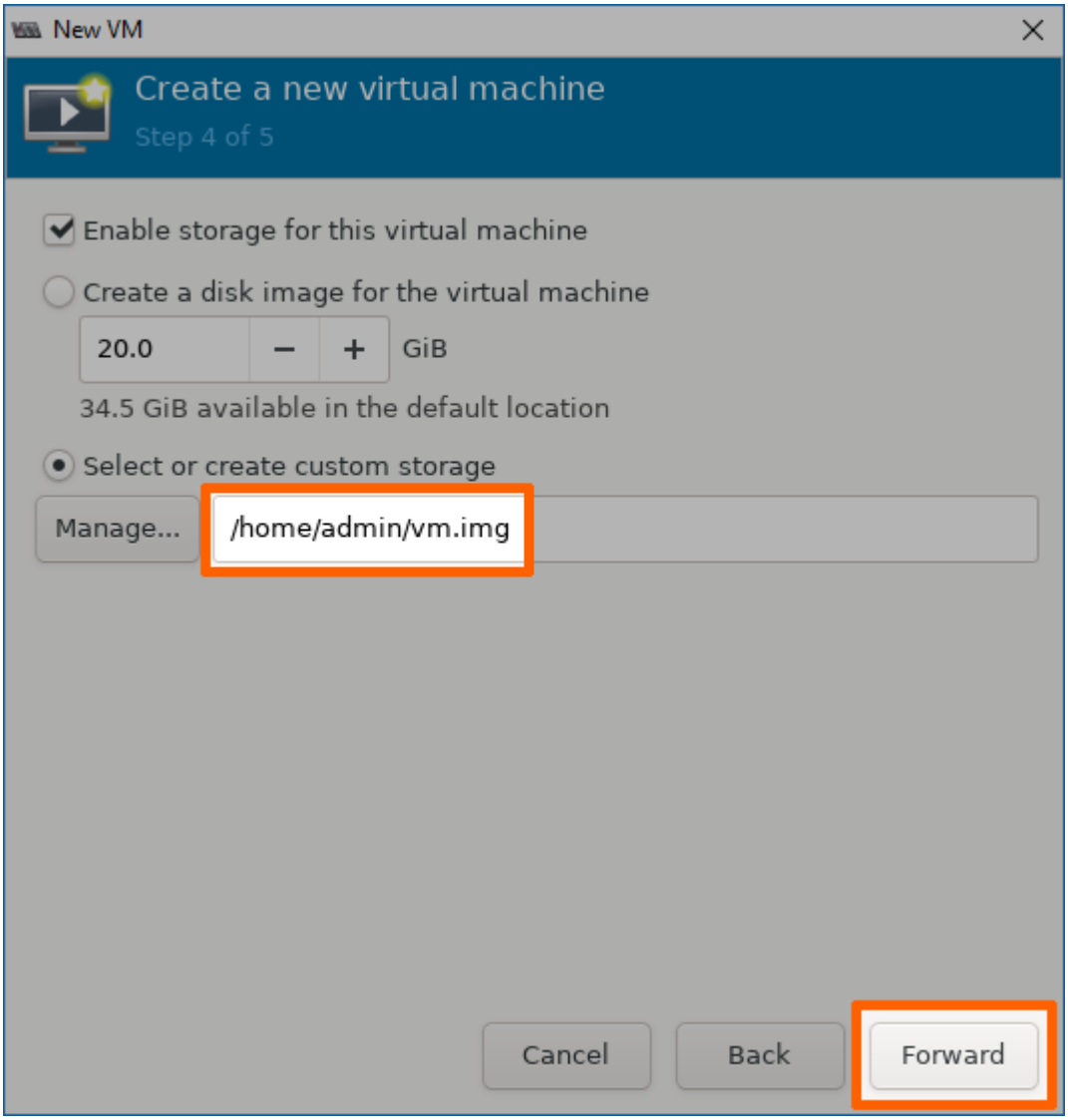
Installing the Operating System

Now the virtual machine will be initiated and the installation of the OS will be started.

1. Select the image that was created in the **Choose Storage Volume** window.
2. Click **Choose Volume**.

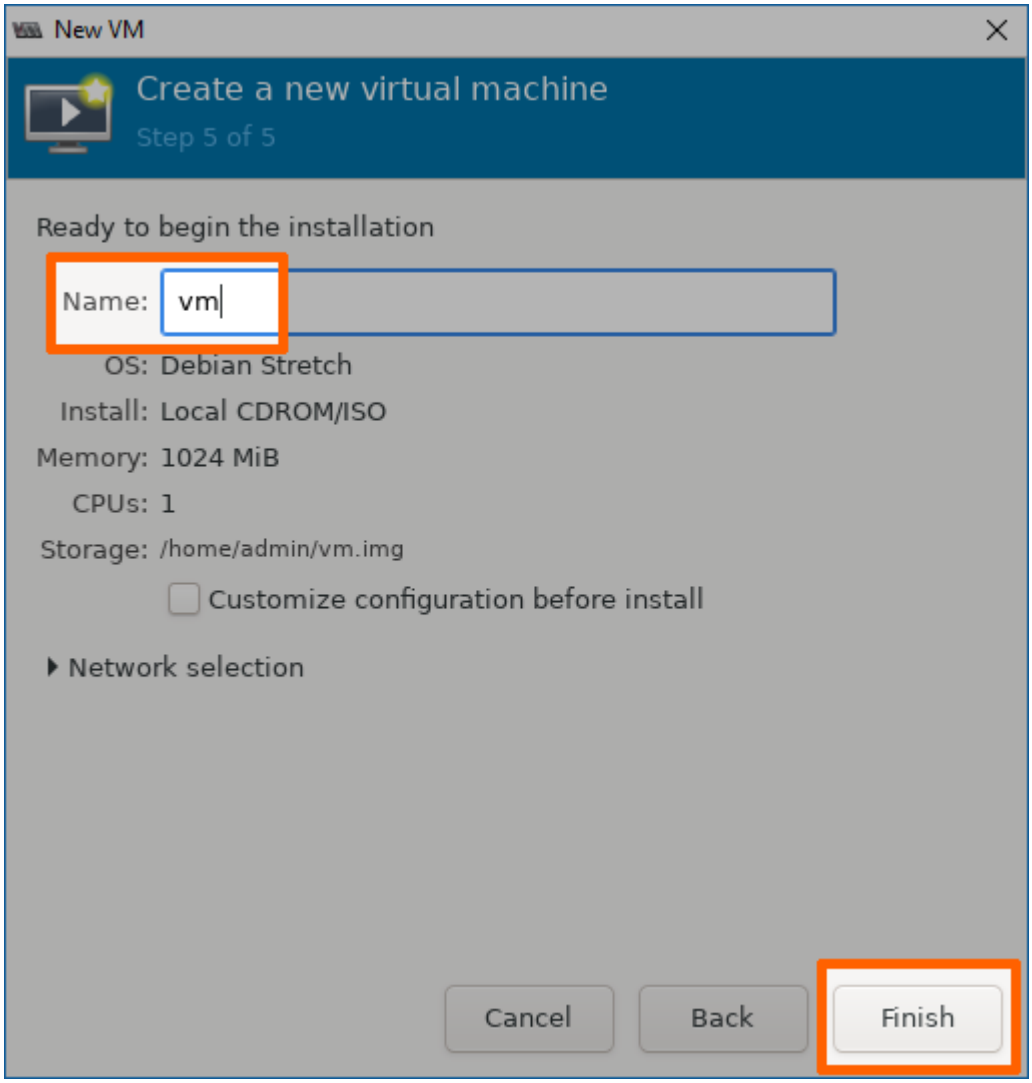


3. Click **Forward** to initialize the installation of the OS on the virtual machine.



4. Enter a name for the virtual machine.

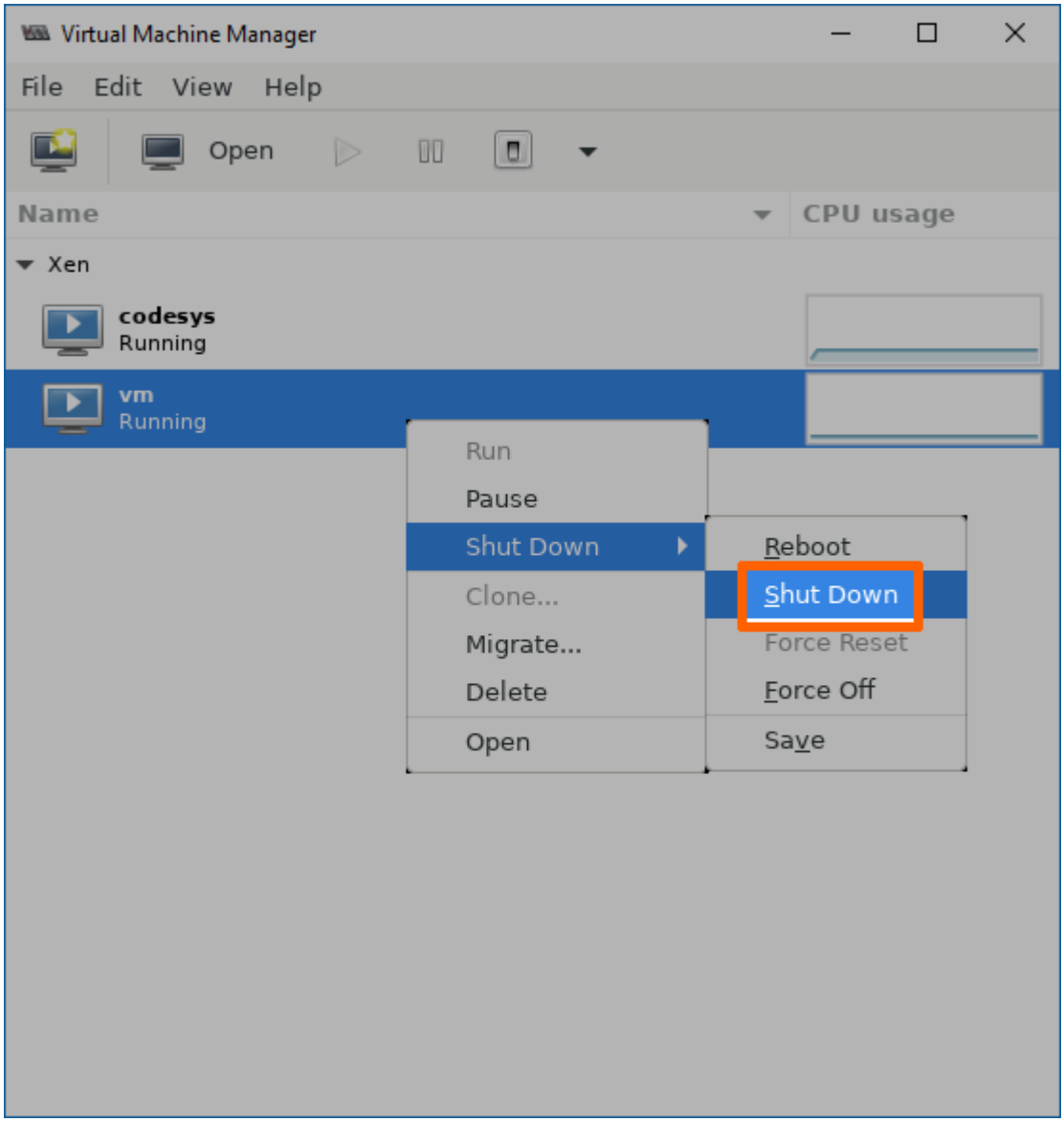
5. Click **Finish**. The virtual machine will be initiated and the installation of the OS will be started.



6. Complete the installation of the OS. Please follow the steps provided by the vendor.

7. After the installation is completed, right-click the virtual machine in the main Virtual Machine Manager window.

8. Select **Shut Down > Shut Down** to shut down the VM.



Copying the IMG File to a Local Workstation

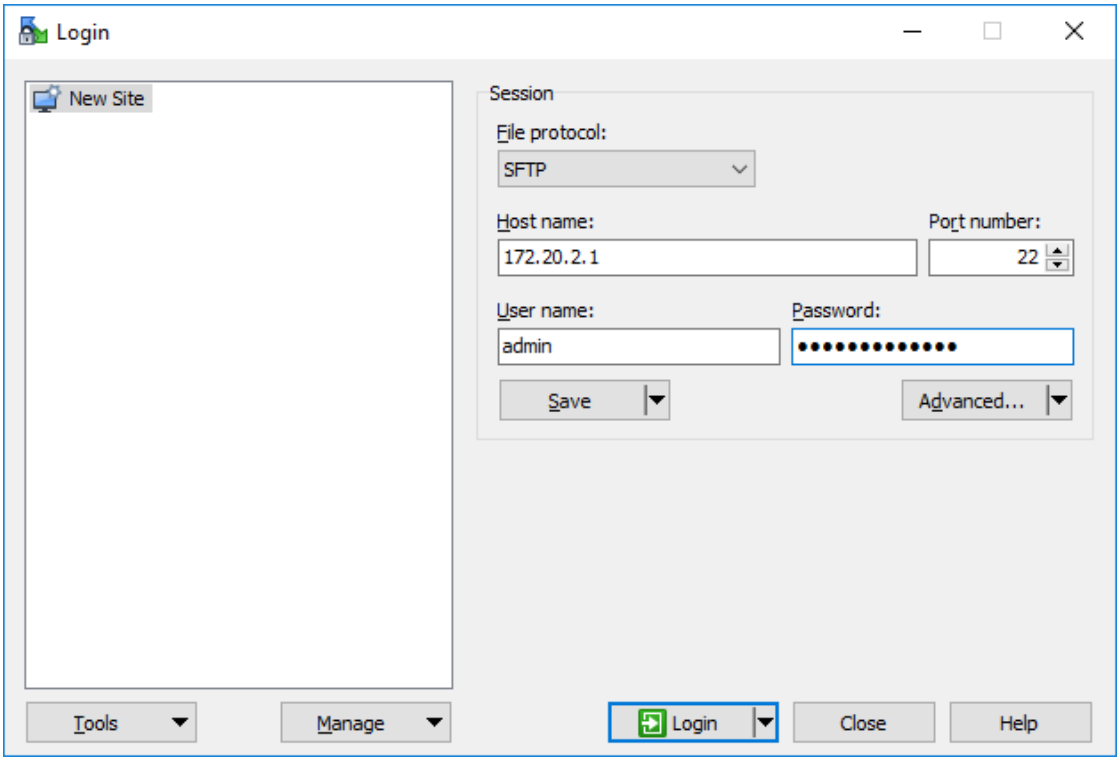
With the generation of the VM, the IMG file of the VM has also been generated on the Nerve Device. Now you can copy the IMG file to your local workstation.

1. Switch to the SSH client window.
2. Enter sudo chmod o+r /home/admin/<yourvm>.img.

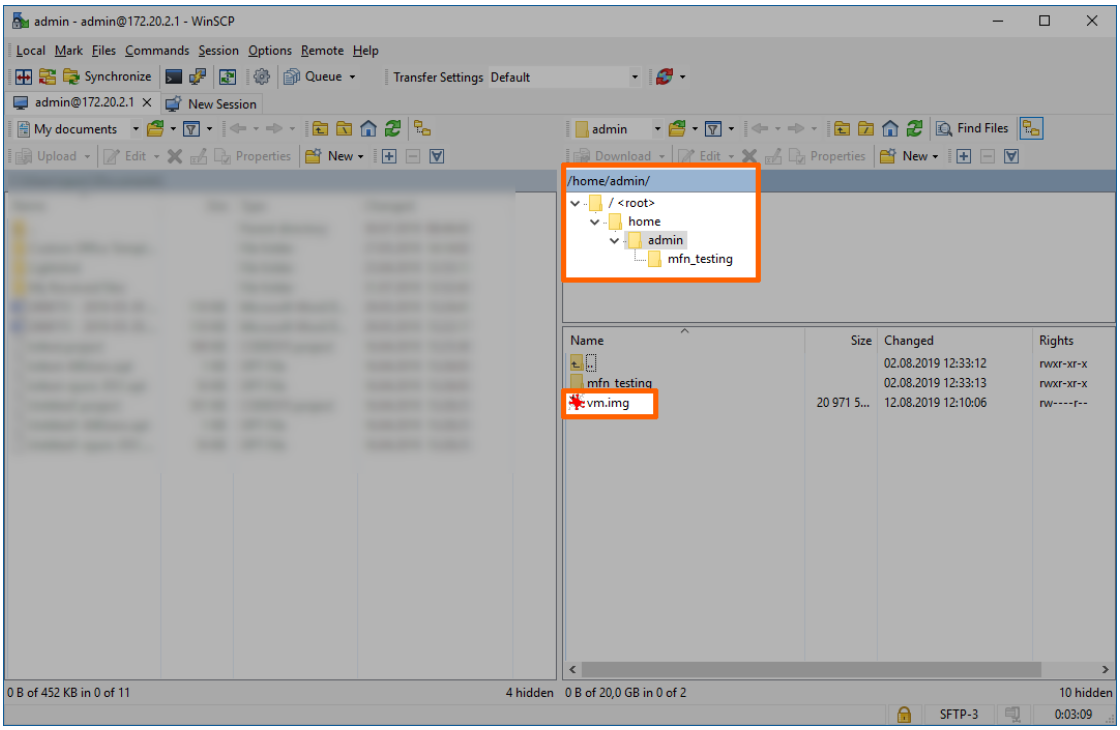
> **NOTE**
>
> The IMG file is called vm.img in this example. Please replace the placeholder name of the image in the command with the name of your IMG file.

3. Open a file transfer client like WinSCP.
4. Enter the IP address for host access of your Nerve Device under **Host Name**.

5. Enter the credentials for host access to the Nerve Device below under **User name** and **Password**.



6. Copy <yourvm>.img to your local workstation.



The virtual machine has now been generated on the Nerve Device and the IMG file of the virtual machine is now on your local workstation.

NOTE

Do not deploy the virtual machine from the process above to the same Nerve Device through the Management System. The virtual machine will be present twice. The deployment of the virtual machine from the process above should be done to different nodes.

Obtaining the XML file

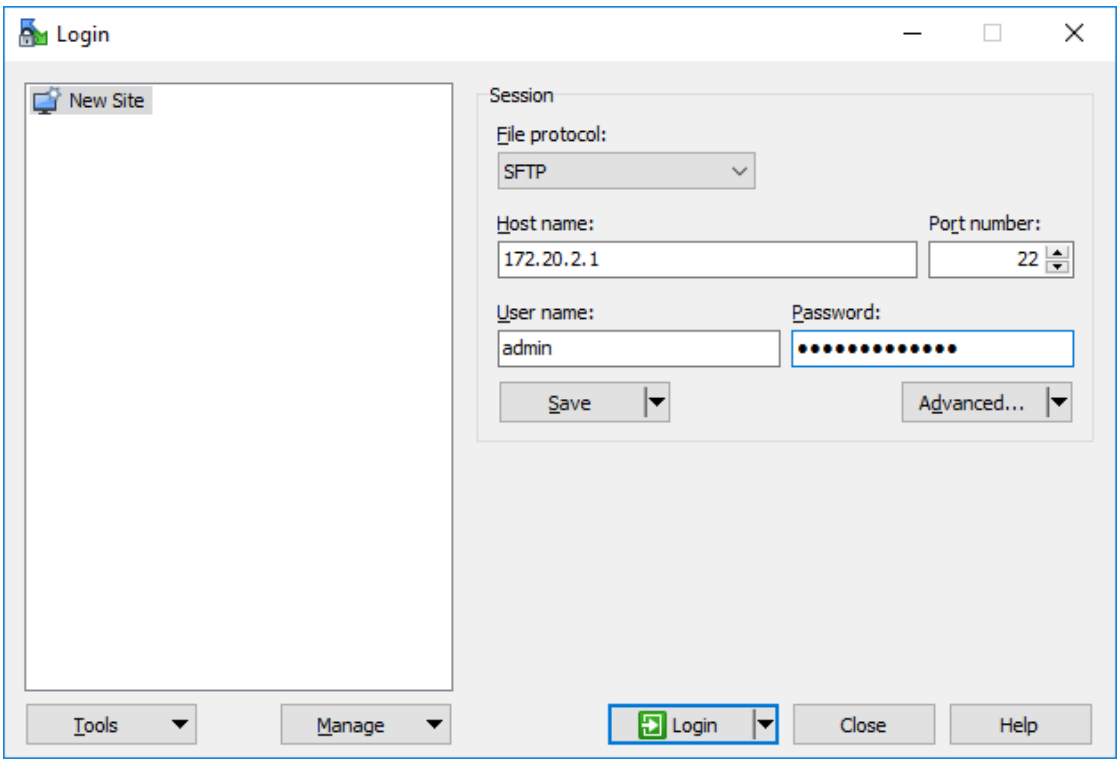
When the IMG was generated on the Nerve Device, an XML file for the IMG file was generated as well. It also has to be obtained manually.

1. Switch to the SSH client window.
2. Enter sudo chmod o+r /etc/libvirt/libxl/<yourvm>.xml.

NOTE

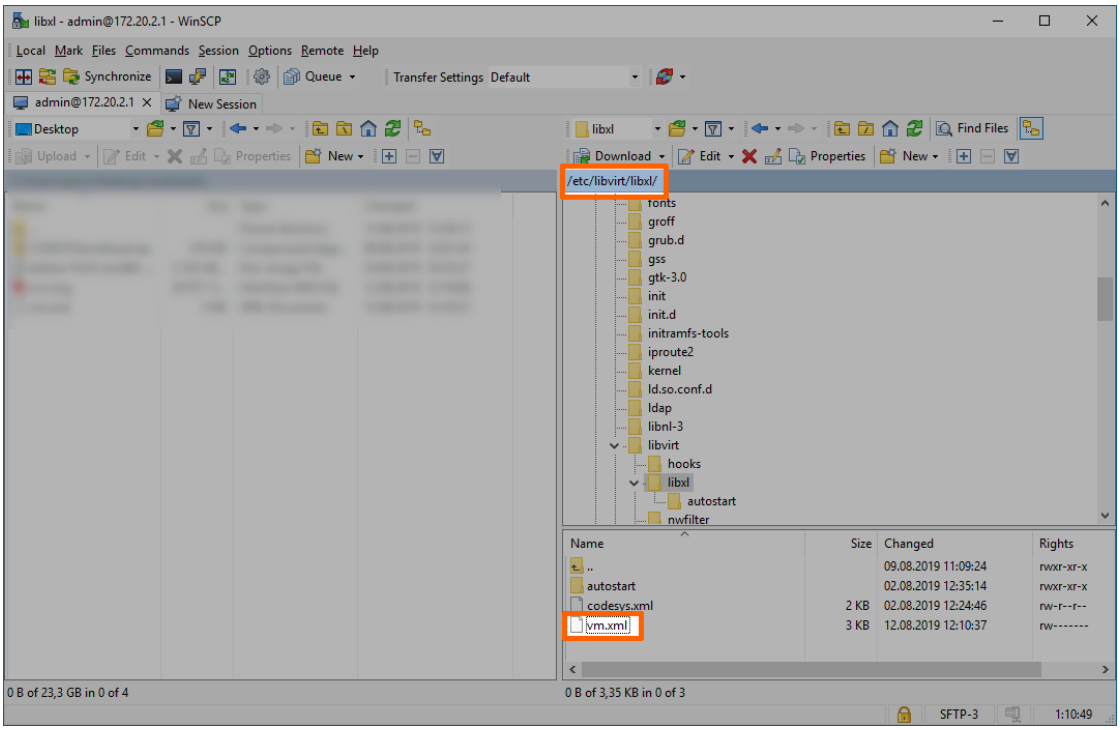
The XML file is called vm.xml in the screenshots of this example. Please replace the placeholder name of the XML file in the command with the name of your XML file.

3. Open a file transfer client like WinSCP.
4. Enter the IP address for host access of your Nerve Device under **Host Name**.
5. Enter the credentials for host access to the Nerve Device below under **User name** and **Password**.



6. Change the path to /etc/libvirt/libxl/ on the Nerve Device. You can find the **etc** directory in the **root** directory.

7. Copy the <yourvm>.xml file to your local workstation.



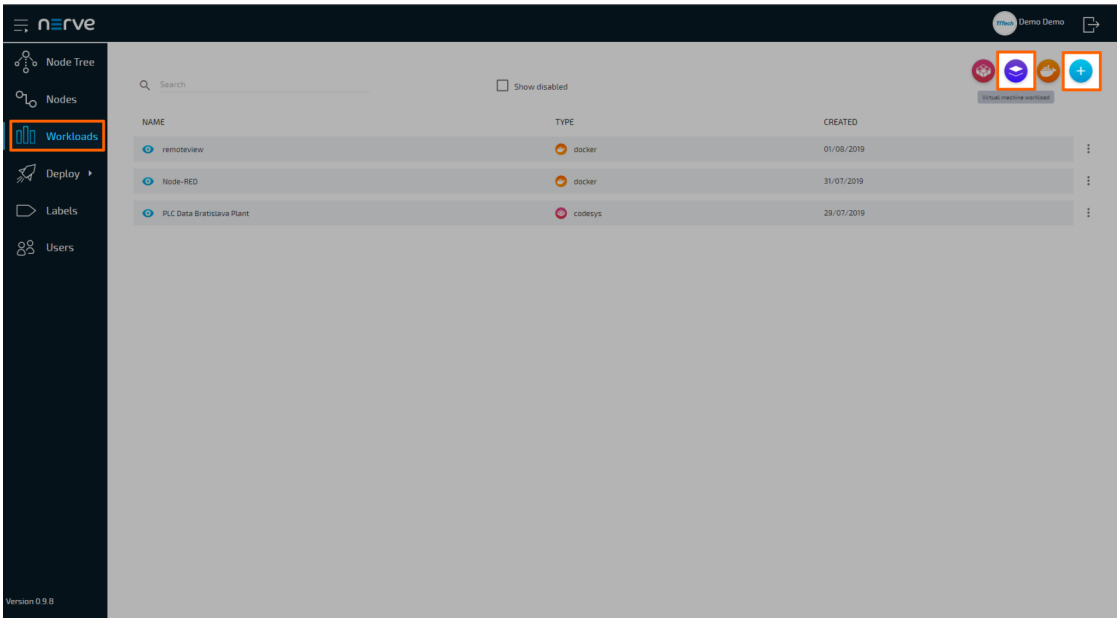
With this you have all the necessary files to provision a Virtual Machine workload for this virtual machine.

Provisioning a Virtual Machine Workload

The following instructions cover the basic requirements for provisioning a Virtual Machine workload. Optional settings will be left out. Extended options are addressed in the last section of this chapter.
There are two further types of workloads that can be provisioned: CODESYS workloads and Docker workloads. The process for each workload is highlighted in its respective chapter.

1. Log in to the Management System.
2. Select **Workloads** in the left-hand menu.
3. Select **+** in the upper-right corner.

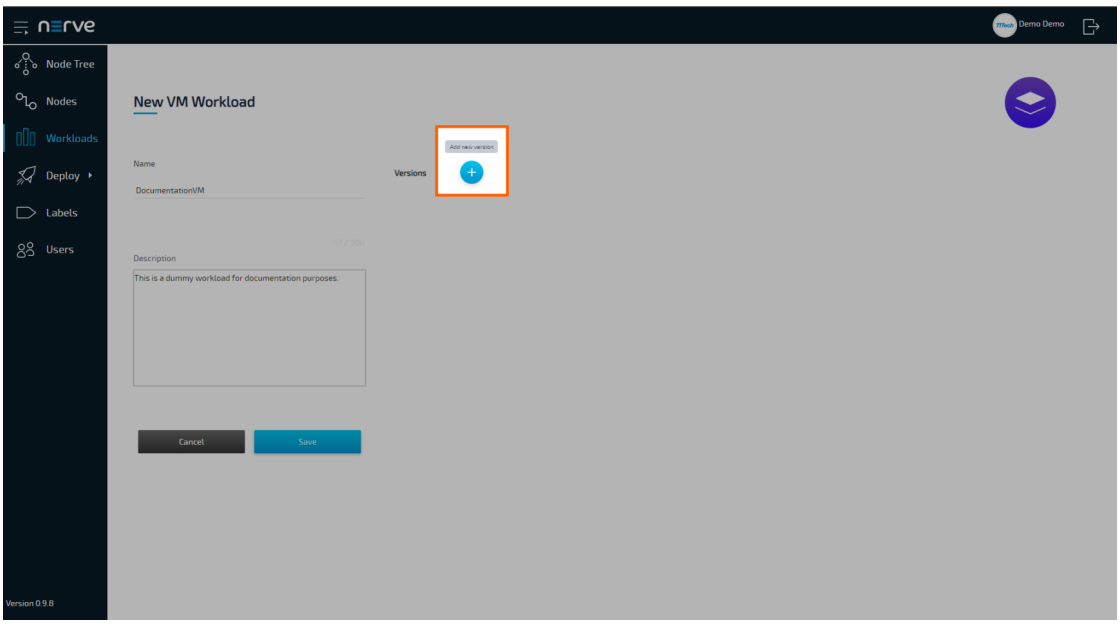
4. Select the virtual machine symbol (**Virtual machine workload**) in the middle of the three symbols that expanded.



5. In the new window, enter a name for your workload.

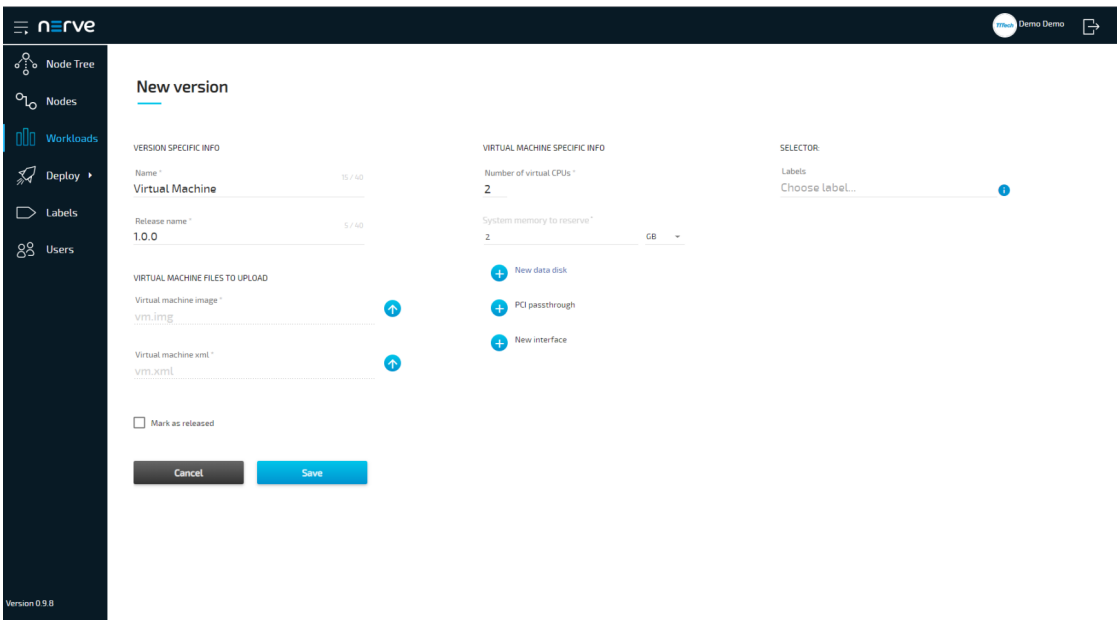
> **NOTE**
>
> Do not use codesys as a name for a Virtual Machine workload. Deploying a VM workload named codesys will interfere with the system.

6. Select **+** next to **Versions** to add a new version of the workload.



7. In the next window, enter the following information:

Item	description
Name	In the new window, enter a **Name** for the version of this workload.
Release name	Enter a **Release name** for the version of this workload.
Virtual machine image	Click the **upward arrow** symbol here to add the virtual machine image. The image has to be an IMG or RAW file.
Virtual machine xml	Click the **upward arrow** symbol here to add the virtual machine XML file.
Number of virtual CPUs	Enter the number of virtual CPUs you would like to use for this virtual machine.
System memory to reserve	Assign how much system memory you would like to reserve for this workload.



8. Select **Save** in the lower-left corner.

The workload has now been provisioned and is ready to be deployed in the **Deploy** menu.

> **NOTE**
>
> While some settings are not required to provision a Virtual Machine workload in the Management System, you will have to fill out additional settings for the workload to perform as desired. Depending on the virtual machine you want to deploy, you may have to define new interfaces. Please keep this in mind and make sure to learn the details about your virtual machine.

Settings for Virtual Machine Workloads

In the instructions above, all optional settings have been left out. Below is an overview of all the options with an explanation to each option.

Setting	Description
VERSION SPECIFIC INFO	**Name** A name for your workload version. This could be a reminder for a certain configuration. Example: "Unlimited" as a name for a virtual machine that has unlimited access to CPU resources. **Release name** A release name for your workload version. This could be a version number. Example: 1.0.1
VIRTUAL MACHINE FILES TO UPLOAD	Two files need to be added here: **Virtual machine image** Upload the virtual machine image with the file extension RAW or IMG here. Do this by clicking the **upward arrow** symbol and selecting the file in the file browser. This is the first of the two files you generated in the process before. **Virtual machine xml** Upload the virtual machine XML file here. Do this by clicking the **upward arrow** symbol and selecting the file in the file browser. This is the second of the two files you generated in the process before. Please be aware that the settings defined under **Virtual machine specific info** are going to overwrite parts of this XML file.

Setting	Description
Virtual machine specific info	**Number of virtual CPUs** Define the number of virtual CPUs you would like to assign to this virtual machine. The CPUs are then reserved exclusively for the Virtual Machine workload and cannot be used by other processes. This setting is mandatory and the workload cannot be provisioned if it is left blank. **System memory to reserve** Assign how much system memory you would like to reserve for this workload. The memory assigned here will be reserved exclusively for this Virtual Machine workload and will not be available for any other processes. This setting is mandatory and the workload cannot be provisioned if it is left blank. **Add new data disk** Click here to add a new data disk for the virtual machine. This data disk functions like an extra hard drive for data separate from the virtual machine. Enter a **Data disk name** and define the **Disk size**. **Add PCI passthrough** Click here to add a PCI passthrough to the virtual machine. Enter the PCI address of the interface you would like to pass through to be directly used by the virtual machine. Please note that the PCIe address is specific to a certain hardware. If you use this option, you should probably limit the installation targets to Nodes with this specific hardware by using selectors. See the labels chapter for more information on selectors. **Add new interface** Click here to add a new interface. You can choose between a bridged interface and a NAT-interface. For NAT interfaces you can define port mappings for TCP and UDP. The names of the interfaces here have to match the names of the pre-defined network interfaces. Also, make sure to not use reserved ports for the workload. Please refer to the networking chapter for more detailed information.
SELECTOR	**Labels** If you have defined labels and assigned them to nodes, you can add them as selectors to the workload. When deploying a workload, the list of nodes will be filtered automatically to the specified label.
Mark as released	Tick this checkbox if you want to mark this workload as released. Once marked as released, the workload cannot be edited anymore.

Provisioning a Docker Workload

The following instructions cover the basic requirements for provisioning a Docker workload. Optional settings will be left out. Extended options are addressed in the last section of this

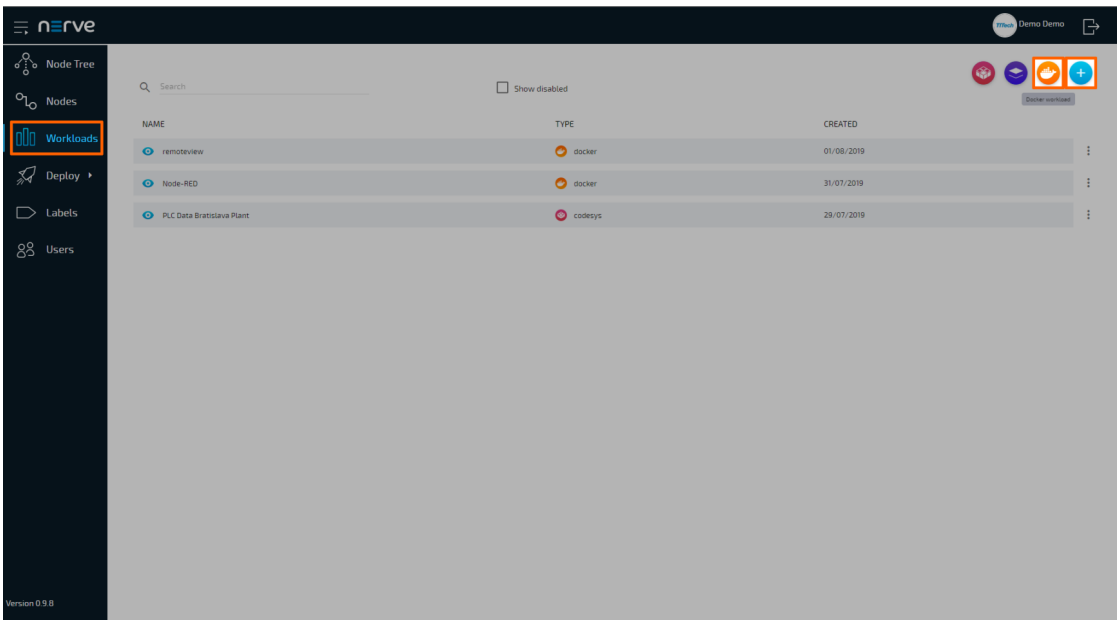
chapter.
There are two further types of workloads that can be provisioned: CODESYS workloads and Virtual Machine workloads. The process for each workload is highlighted in its respective chapter.

NOTE

A Docker image is required for the following instructions. Consult the Docker documentation for help on creating a Docker image.

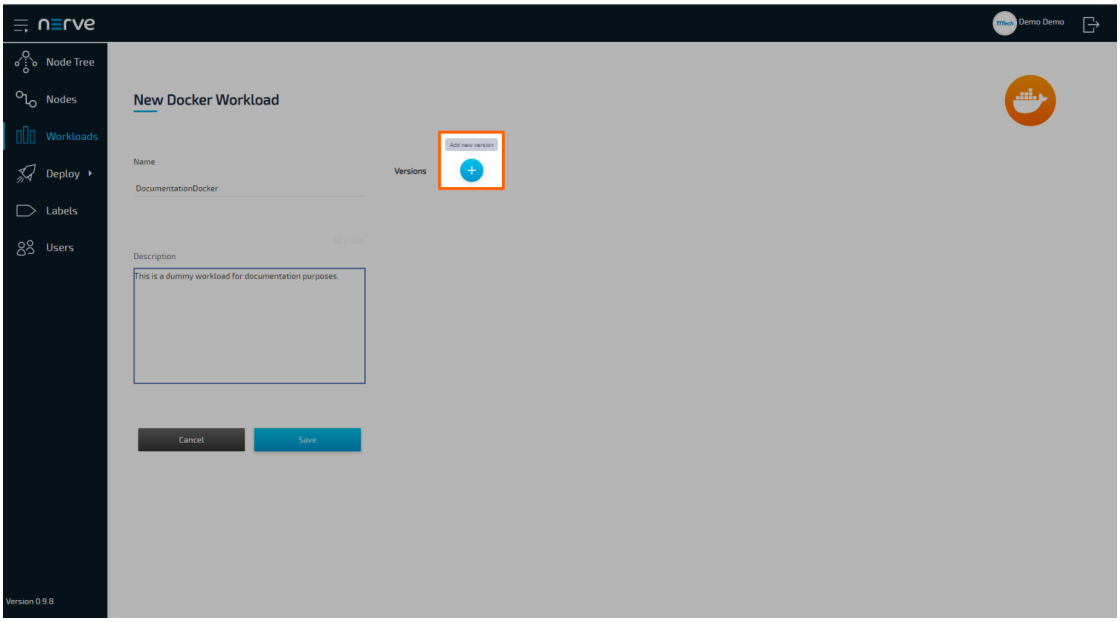
1. Log in to the Management System.
2. Select **Workloads** in the left-hand menu.
3. Select **+** in the upper-right corner.

4. Select the Docker symbol (**Docker workload**) on the right of the three symbols that expanded.



5. In the new window, enter a name for your workload.

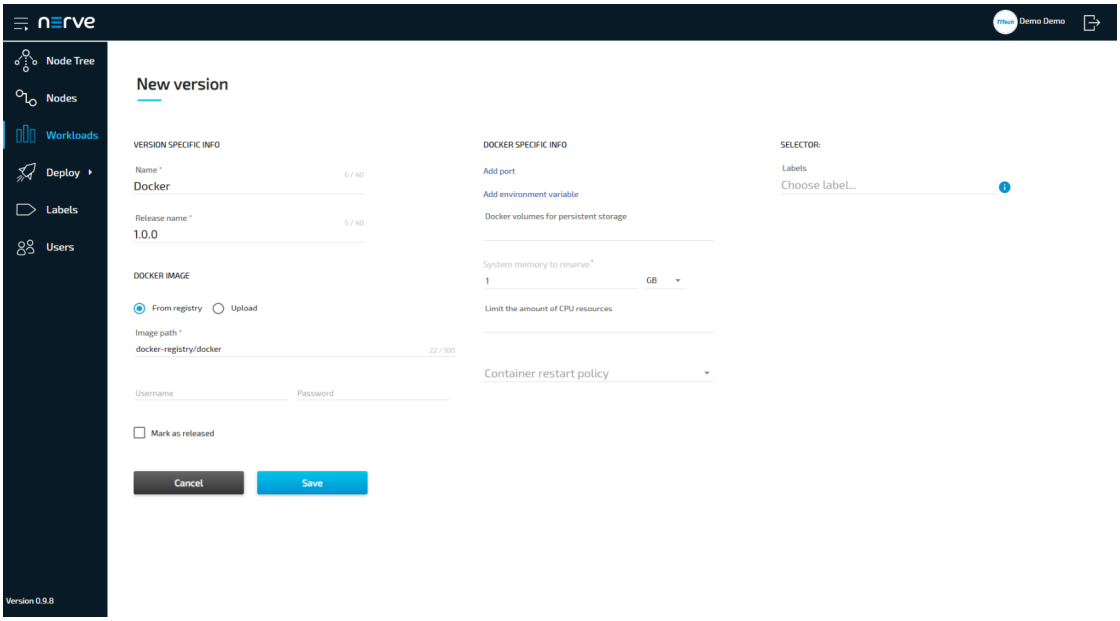
6. Select **+** next to **Versions** to add a new version of the workload.



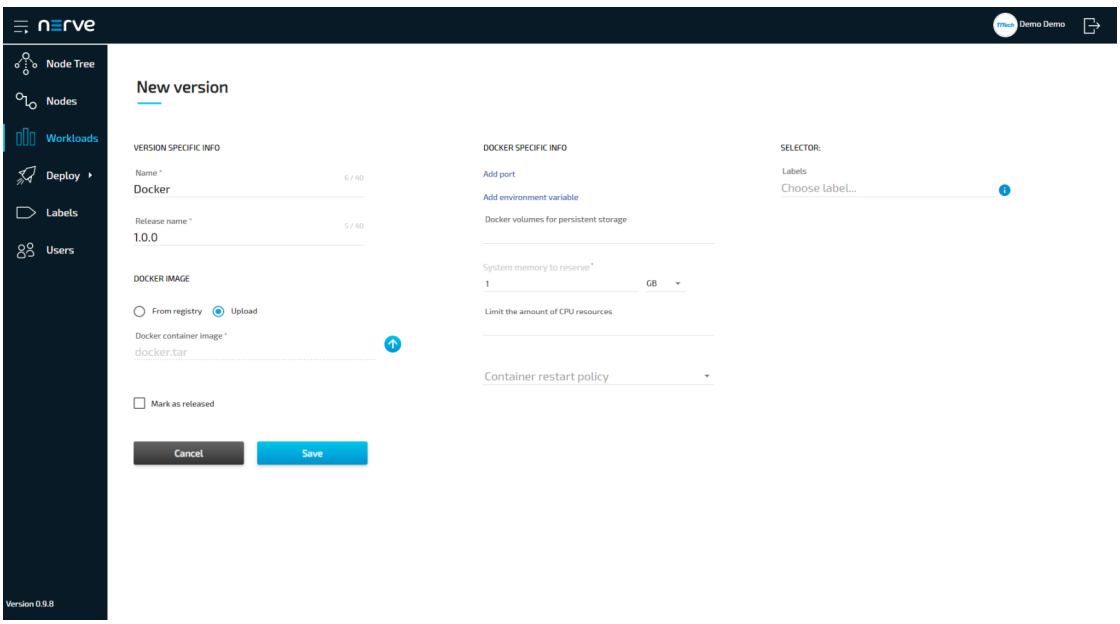
7. In the next window, enter the following information:

Item	Description
Name	Enter a **Name** for the version of this workload.
Release name	Enter a **Release name** for the version of this workload.
DOCKER IMAGE	Select **From registry** or **Upload**. If you select **From registry**, you need to enter the Docker registry to the Docker container image under **Image path**. If you select **Upload**, click the **upward arrow** symbol to upload the Docker container image.
System memory to reserve	Assign how much system memory you would like to reserve for this workload.

From registry:



Upload:



8. Select **Save** in the lower-left corner.

The workload has now been provisioned and is ready to be deployed in the **Deploy** menu.

> **NOTE**
>
> While some settings are not required to provision a Docker workload in the Management System, you will have to fill out additional settings for the workload to perform as desired. Depending on the Docker container you want to deploy, you will need to define ports and configure environment variables. Please keep this in mind and make sure to learn the details about your Docker container.

Settings for Docker Workloads

In the instructions above, all optional settings have been left out. Below is an overview of all the options with an explanation to each option.

Setting	Description
VERSION SPECIFIC INFO	**Name** A name for your workload version. This could be a reminder for a certain configuration. Example: "Unlimited" as a name for a Node-RED version that has unlimited access to CPU resources. **Release name** A release name for your workload version. This could be a version number. Example: 1.0.1
DOCKER CONTAINER FILE MANAGEMENT	You can select between two options here and either use a Docker registry URL to link to an online repository or upload the Docker container image from your workstation. **From registry** If you select this, you have to specify a URL pointing to the Docker container image under **Image path** below. Please note the differences between public Docker Hub registries and private registries. Private registries require the full URL to be specified, as well as a username and password if they require authentication. Public Docker Hub registries can be specified in their short form. Examples: • Public Docker Hub registry nodered/node-red-docker • Private registry with authentication and a tag at the end auth.docker.test.host.cloud/workload:v1.3 **Upload** If you select this, you have to upload the Docker container image from your workstation. Do so by clicking the **upward arrow** symbol to open your file browser.

Setting	Description
DOCKER CONTAINER SPECIFIC INFO	**Add port** Click here to define a **Host port** and a **Container port**. Please make sure to select the appropriate container port as it depends on the Docker container you are deploying. Also, make sure to not use reserved ports for the workload. Please refer to the networking chapter for more detailed information. **Add environment variable** Click here to add an environment variable (**Env. variable**) and its **Variable value**. Please make sure to define the appropriate variables and values as they depend on the Docker container you are deploying, **Docker volumes for persistent storage** If your workload needs persistent storage, specify the path of the Docker volumes in the Docker container here. The volume for persistent storage is not erased even if the Docker workload is undeployed. **System memory to reserve** Assign how much system memory you would like to reserve for this workload. The memory assigned here is an upper limit that the Docker workload can use and is not exclusively reserved for the Docker workload. Other processes can use these resources as well. **Limit the amount of CPU resources** Specify here how many CPU cores the workload is allowed to use. If this field is left blank, the workload is allowed to use all available resources. **Container restart policy** Choose the container restart policy here to determine when the Docker container can be restarted. • **no** The container does not restart automatically. • **on-failure** The container restarts when it exits due to an error. • **always** The container restarts every time it stops. However, manually stopping the container is the exception. If a container is manually stopped, it is only restarted when the Docker daemon restarts or the container is restarted manually. • **unless-stopped** The container only restarts if it is manually stopped.
SELECTOR	**Labels** If you have defined labels and assigned them to nodes, you can add them as selectors to the workload. When deploying a workload, the list of nodes will be filtered automatically to the specified label.

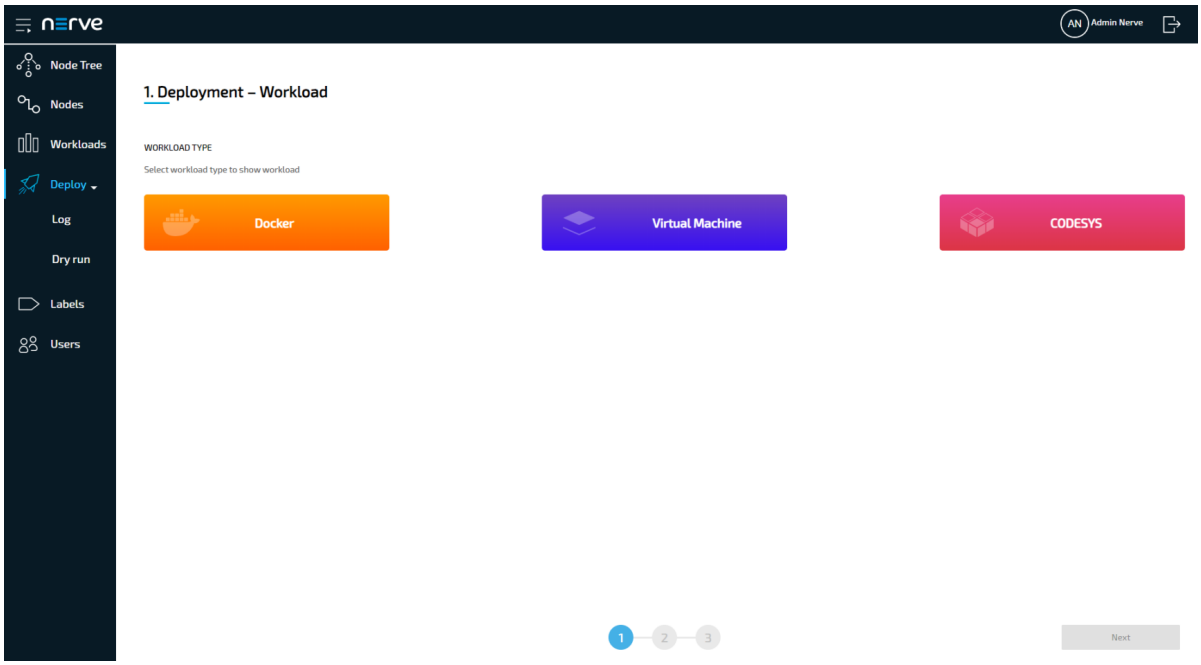
Setting	Description
Mark as released	Tick this checkbox if you want to mark this workload as released. Once marked as released, the workload cannot be edited anymore.

NOTE

UDP port mapping is not available for Docker containers.

Deploy Menu

Workloads that have been provisioned in the Management System are ready to be deployed to nodes. However, before tackling how to deploy workloads to nodes, let's take a look at the **Deploy** menu. It offers three sub-menus:



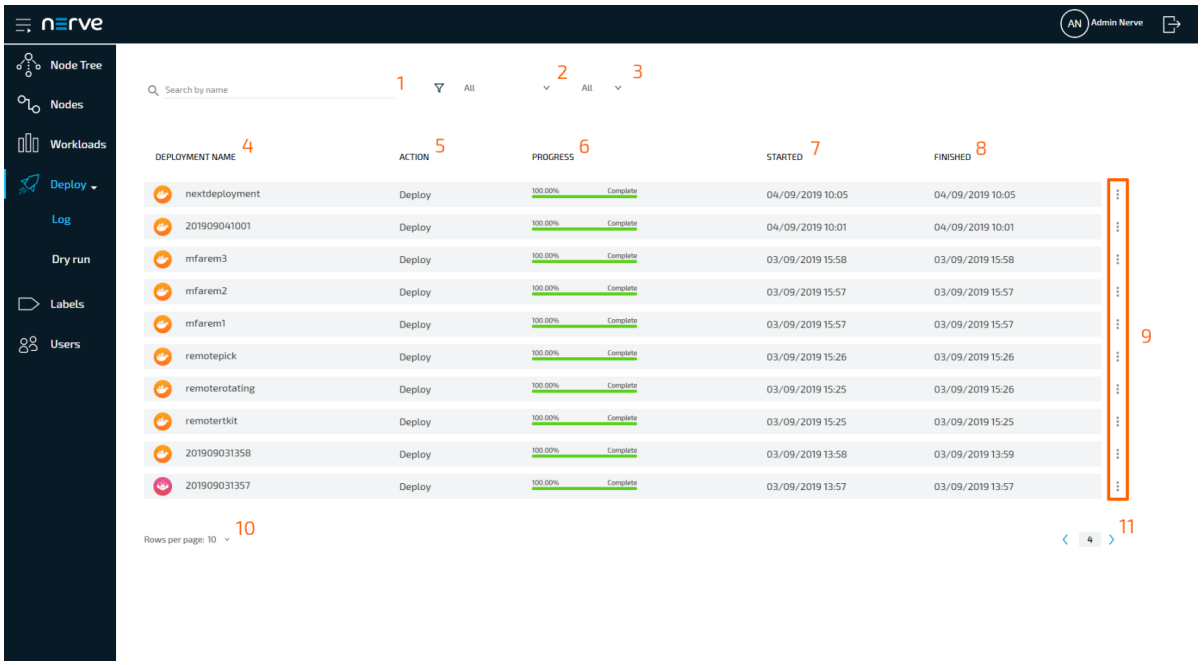
Item	Description
Deploy	This is the landing page of the deploy menu. You can deploy workloads to nodes from here.
Log	This is the history of deployments and dry runs.
Dry run	Structurally the same as the deployment process for workloads, you can simulate the deployment of a workload from this menu.

Log

The log is the history of deployments and dry runs. This includes:

- deployments in progress
- dry runs in progress
- failed deployments
- failed dry runs
- successful deployments
- successful dry runs

It is displayed in reverse chronological order and can be filtered according to search criteria. It also offers some control functionality for active and failed deployments.

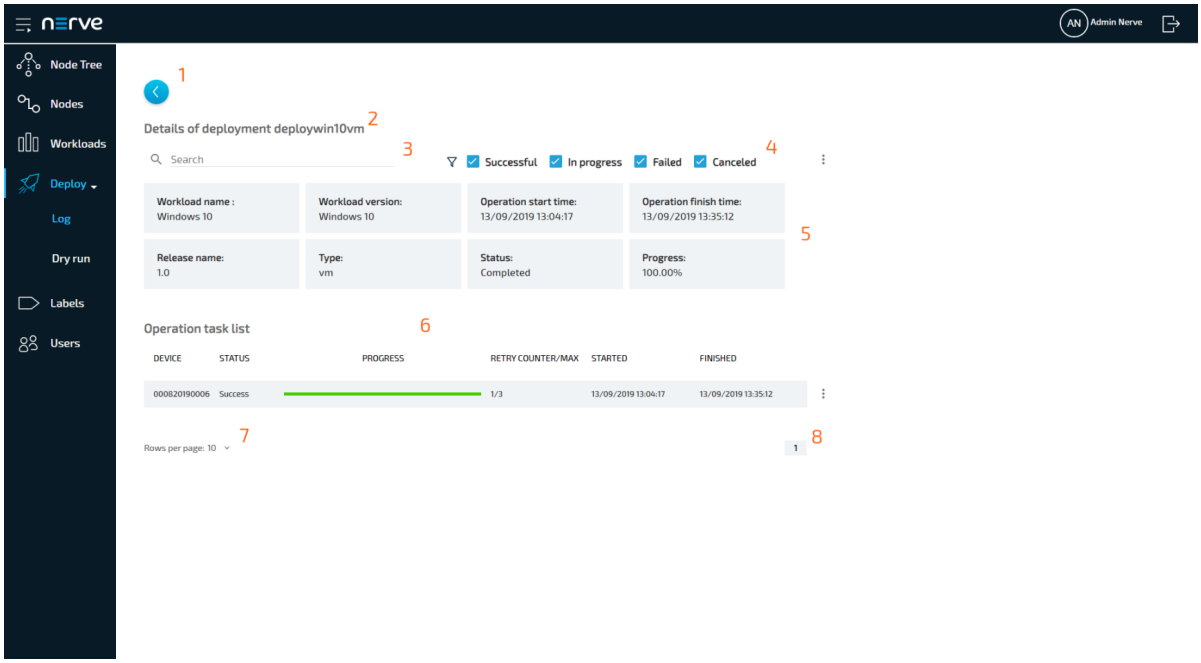


Item	Description
Search by name (1)	Enter text here to filter the list by deployment name.
Deployment Type (2)	Select an option from the drop-down menu to filter the list for **Deploys** or **Dry runs**.
Workload Type (3)	Select an option from the drop-down menu to filter the list for a specific workload type: **VM**, **Docker** or **CODESYS**.
DEPLOYMENT NAME (4)	This is the name of the deployment you have defined in the deployment process. Please note that this is not the name of the workload.
ACTION (5)	Here the deployment type is displayed: **Deploy** or **Dry run**.

Item	Description
PROGRESS (6)	The progress bar is an indicator for both progress and status. Depending on the status of the workload it changes its color: • **Green** If a workload was deployed successfully, the bar will be green at a 100%. • **Blue** If a workload is currently being deployed, the bar will be blue, fill up gradually and display the progress of the deployment in percent. • **Red** If the deployment of a workload has failed, the bar will be red at a 100%.
STARTED (7)	This is the date and time the workload deployment was started. The date format is DD/MM/YYYY.
FINISHED (8)	This is the date and time the workload deployment was completed. The date format is DD/MM/YYYY. This field also shows if a workload is in progress of being deployed. In that case it will show **In progress.**
Ellipsis menu (9)	Select the ellipsis menu to trigger an overlay with the **DELETE** option. Selecting **DELETE** will remove the entry from the log. When a workload is in progress, this symbol is grayed out.
Rows per page (10)	Specify how many log entries are displayed on one page. You can select **5**, **10** or **15** workloads per page.
Page navigation (11)	Use the arrows to switch between pages. Clicking the number in the middle opens a list of all page numbers. Selecting a number jumps to that page.

Deployment Details

Clicking an entry in the log will show you the details of the deployment.



Item	Description
Back button (1)	Click here to return to the log.
Header (2)	The header states the name of the deployment. Details of deployment <deploymentname>.
Search (3)	Enter text here to filter the **Operation task list** by device name. The search function can be combined with the status checkboxes to the right of the search bar.
Status checkboxes (4)	The checkboxes to the right of the search bar filter the **Operation task list** by status: • **Success** Deployments that were completed successfully are shown in the list if this checkbox is ticked. • **In progress** Deployments that are currently in progress are shown in the list if this checkbox is ticked. • **Failed** Deployments that could not be completed are shown if this checkbox is ticked. • **Canceled** Deployments that have been aborted by the user are shown in the list if this checkbox is ticked. However, it is not possible to cancel deployments in this version. If a checkbox is ticked, tasks with the corresponding status will be displayed in the operation task list. All checkboxes are ticked by default.

Item	Description
Deployment information (5)	This is information about the deployment. • **Workload name** The name of the workload that has been defined in the provisioning process. • **Workload version** The name of the workload version. • **Operation start time** This is the date and time the workload deployment was started. The date format is DD/MM/YYYY. • **Operation finish time** This is the date and time the workload deployment finished. The date format is DD/MM/YYYY. • **Release name** This is the release name of the workload version. • **Type** This is the type of the workload that has been deployed: **codesys**, **vm** or **docker**. • **Status** This is the status of the deployment. Possible statuses are **Created**, **In progress**, **Completed**, **Error**, **Canceled**. • **Progress** The progress of the deployment in percent.

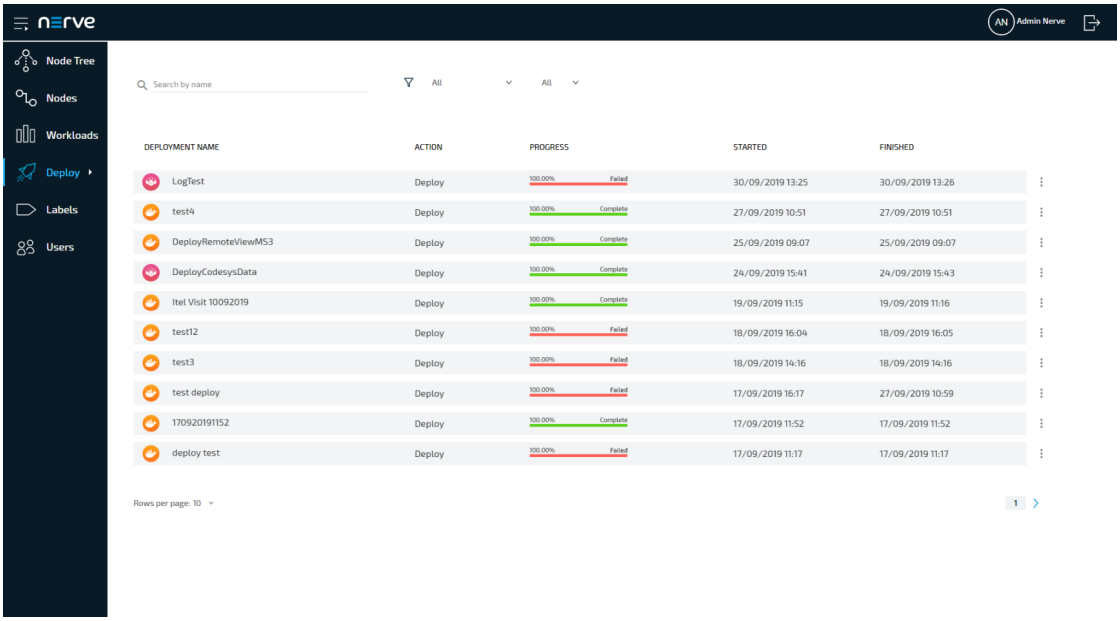
Item	Description
Operation task list (6)	The operation task list displays every deployment that is part of the deployment. The list displays information in six columns: • **DEVICE** This is the serial number of the node that is the target of the deployment. • **STATUS** This is the status of the deployment. The information in this column here corresponds with the checkboxes to the right of the search bar: **Success**, **In progress**, **Failed** and **Canceled** • **PROGRESS** This is the progress bar. It displays a different color and gradually fills up according to the progress and status of the deployment. **Success** If a workload was deployed successfully, the bar will be green at a 100%. **In progress** If a workload is currently being deployed, the bar will be blue and display the progress of the deployment in percent. **Failed** If the deployment of a workload has failed, the bar will be red at a 100%. • **RETRY COUNTER/MAX** In case of failure, the Management System will attempt the deployment of a workload up to three times automatically. The left number of the counter shows the number of the current attempt. The right number is the maximum number of attempts. • **STARTED** This is the date and time the workload deployment was started. The date format is DD/MM/YYYY. • **FINISHED** This is the date and time the workload deployment was finished. The date format is DD/MM/YYYY.
Rows per page (7)	Specify how many deployments are displayed on one page. You can select **5**, **10** or **15** deployments per page.
Page navigation (8)	Use the arrows to switch between pages. Clicking the number in the middle opens a list of all page numbers. Selecting a number jumps to that page.

Restarting a Failed Deployment

In case a deployment fails, the Management System will attempt the deployment of a workload up to three times automatically. After that, the deployment can be restarted manually through the ellipsis menu in the operation task list.

1. Select **Deploy > Log** from the menu on the left.

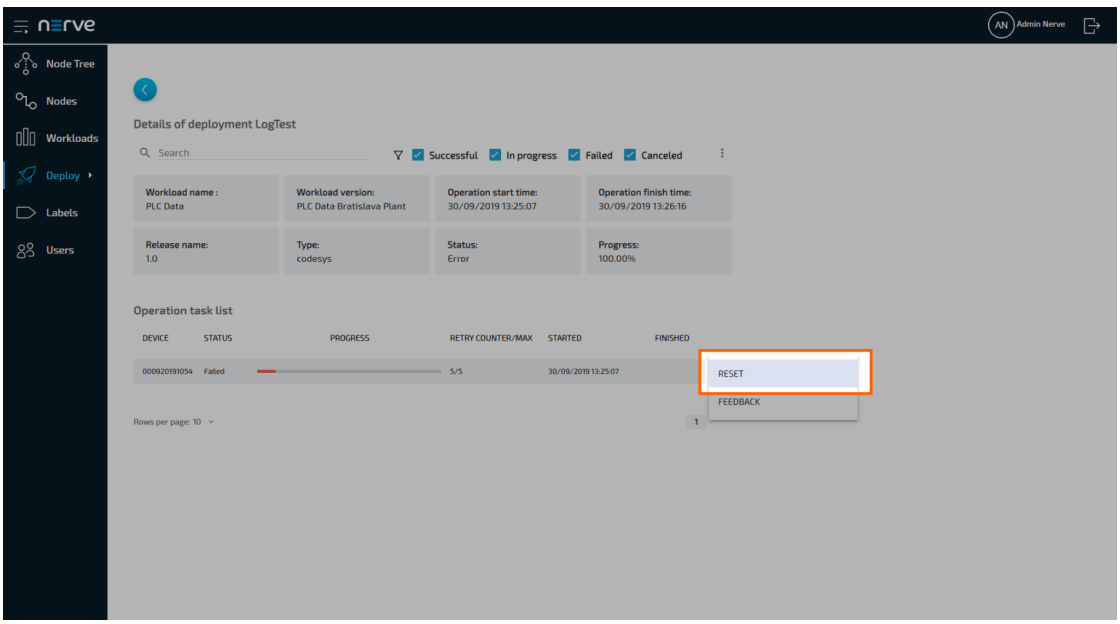
2. Select the failed deployment from the log.



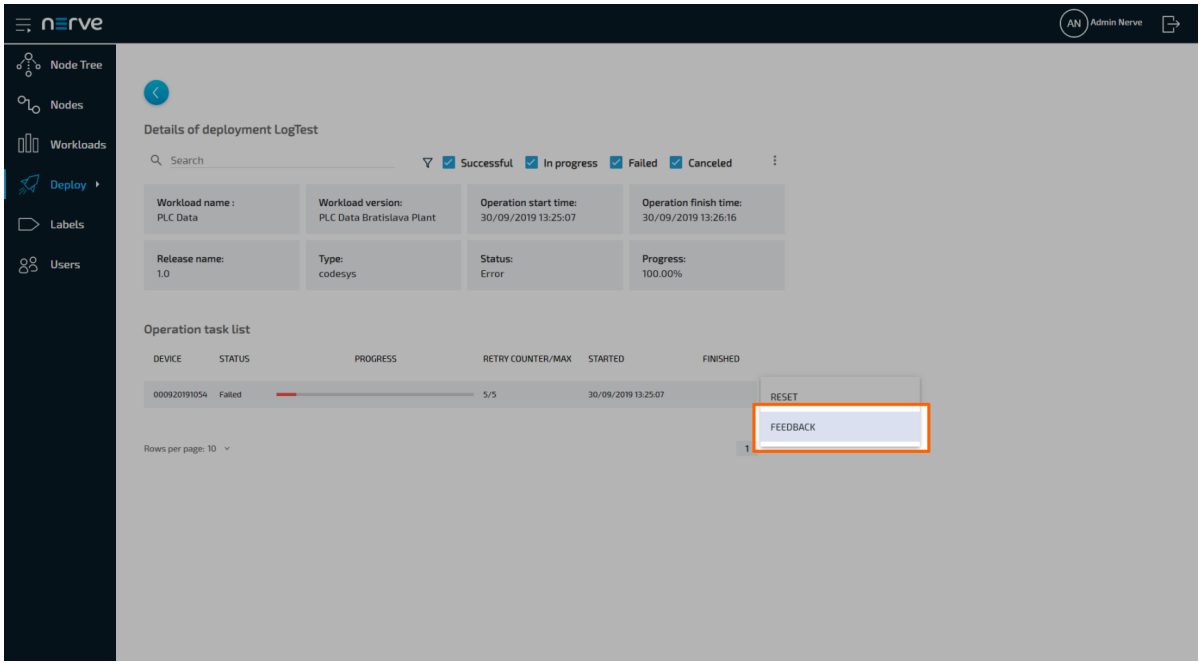
3. Choose the failed deployment from the operation task list.

4. Select the ellipsis menu to the right of the deployment entry.

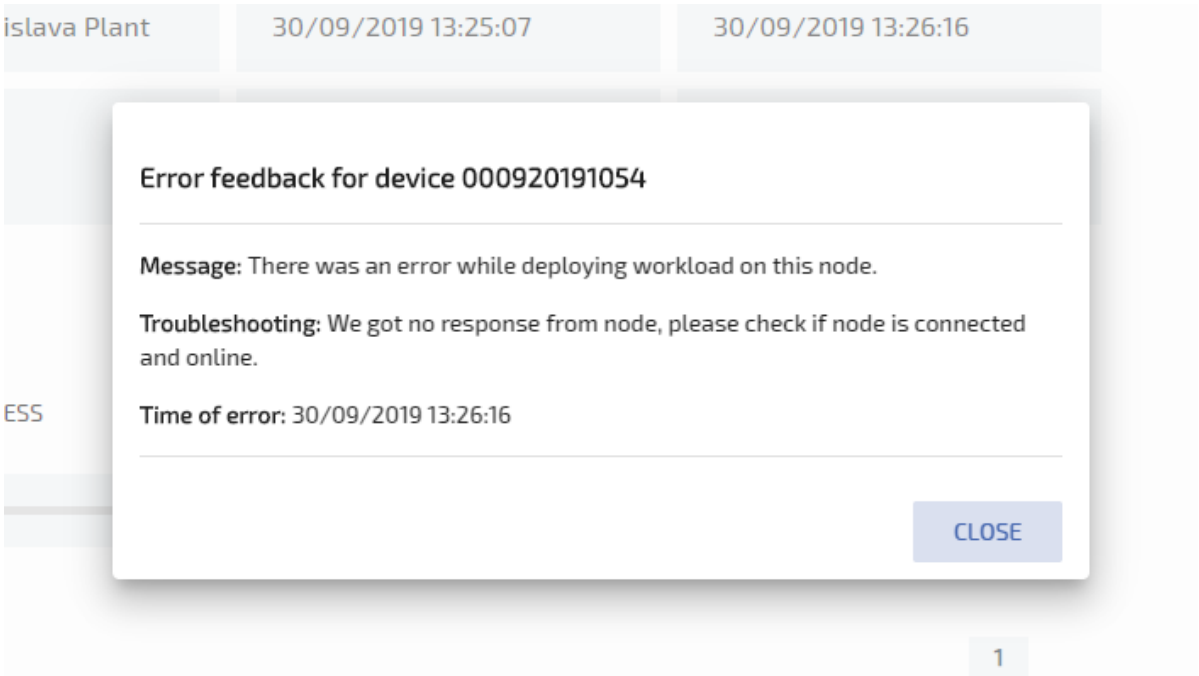
5. Select **RESET** in the overlay that appeared.



The deployment is restarted immediately. If you want to see error information of the deployment, select **FEEDBACK** in the overlay of the ellipsis menu instead.

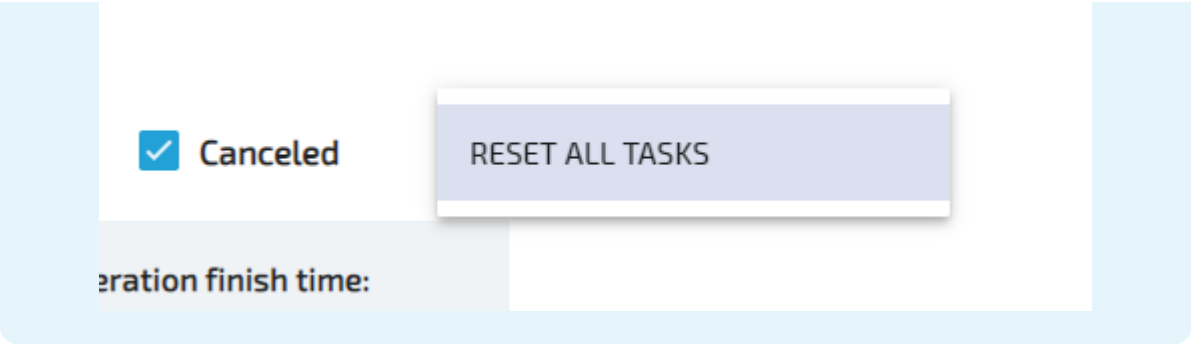


This opens a window giving information on the type of error in a message, a short troubleshooting hint and the time the error occurred. Please note that troubleshooting hints are not available for every feedback message.



NOTE

If you would like to restart all deployments at once, click the ellipsis menu next to the status checkboxes and select **RESET ALL TASKS** from the overlay that appears.



Deploying Workloads and Dry Runs

Deployment of workloads and dry runs are covered in a separate chapter: Deploying a Workload.

Deploying a Workload

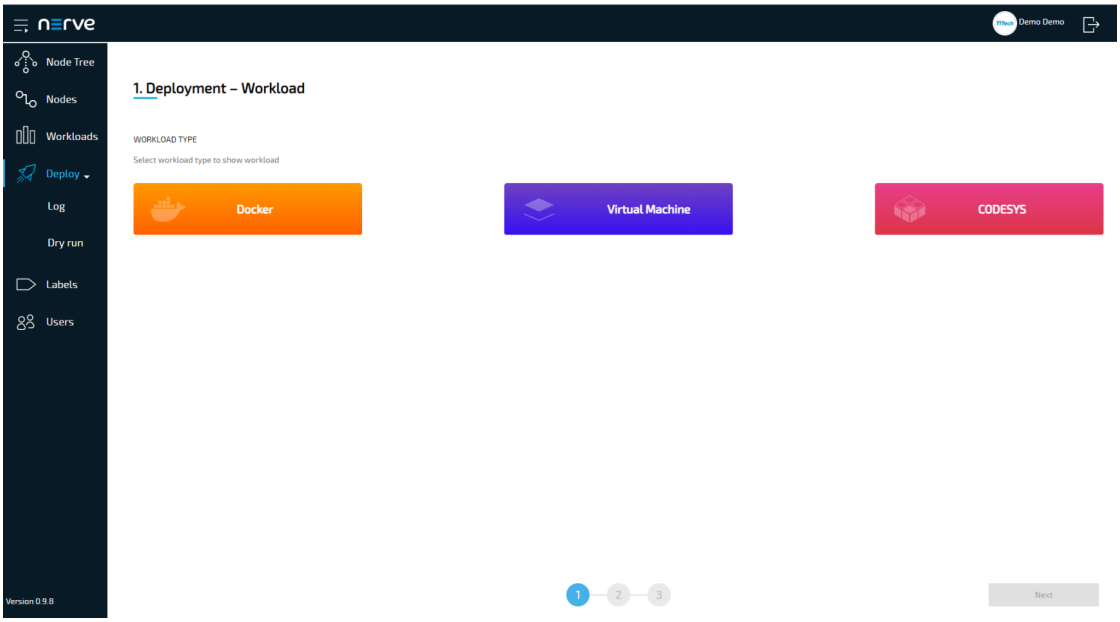
This chapter will take you through the deployment process of the available workloads: CODESYS workloads, Virtual Machine workloads and Docker workloads. The process of deploying workloads is identical for all three types of workloads. Therefore, the instructions below contain no specific information.

NOTE

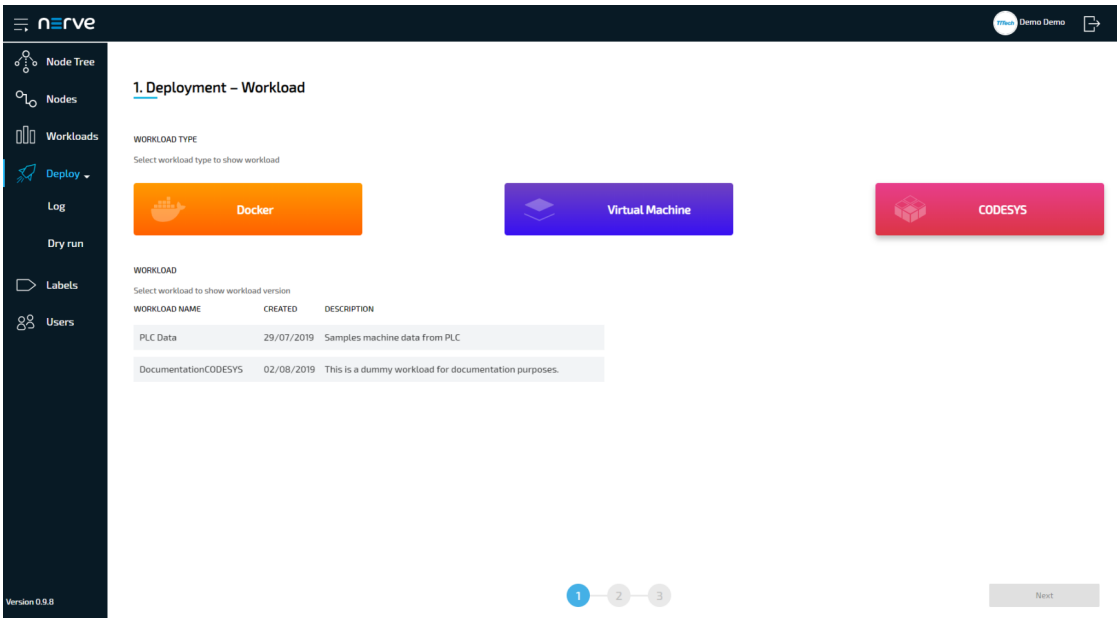
This chapter guides you through deploying a workload. If you want to test a deployment first, select **Deploy > Dry run** in the left-hand menu and follow the steps below starting from step 3.

However, please note that a successful dry run does not guarantee a successful deployment as it is only a simulation.

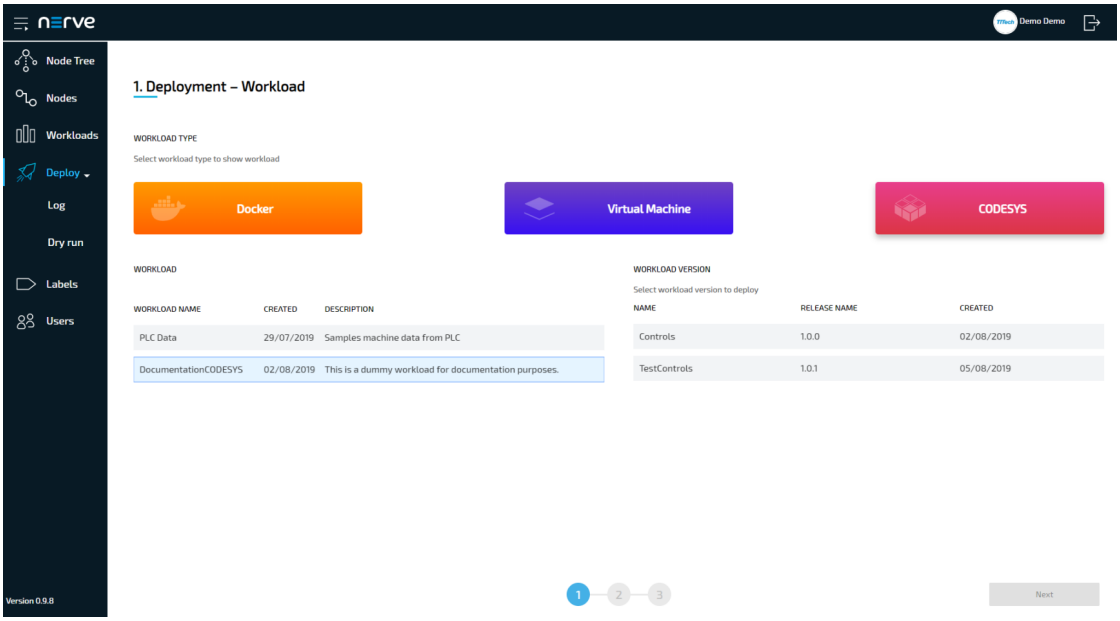
1. Select **Deploy** in the left-hand menu.



2. Select one of the three icons for workload types. A list of corresponding workloads will appear below.

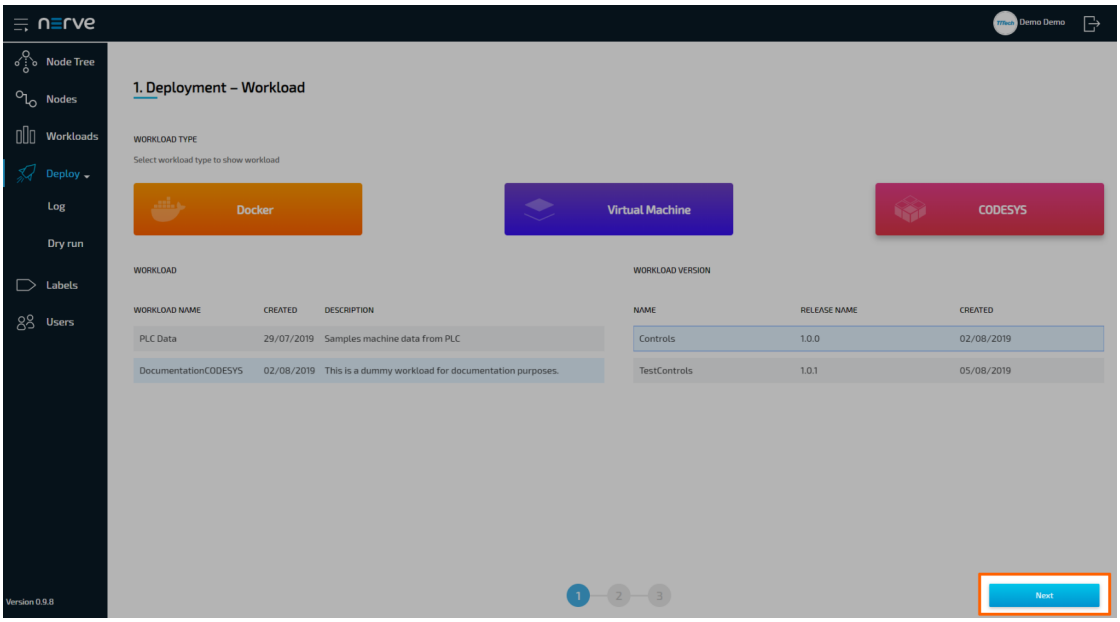


3. Select a workload from the list. A list of versions of this workload will appear to the right.



4. Select the version of the workload you would like to deploy.

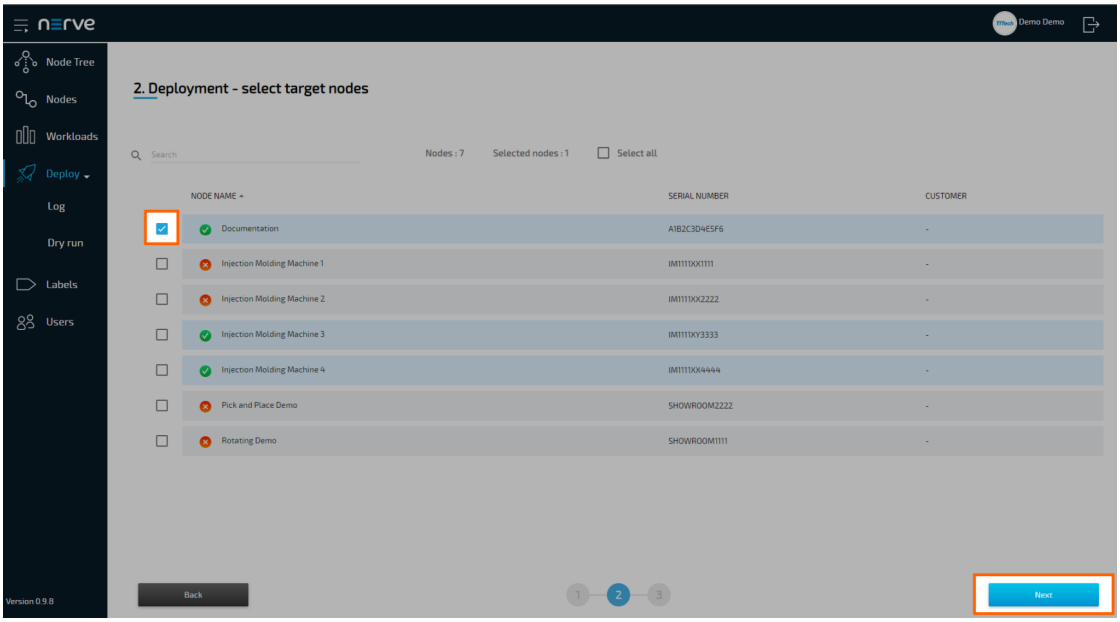
5. Click **Next** in the bottom-right corner.



6. In the next window, select one or more nodes from the list for deployment by ticking the checkboxes on the left.

> **NOTE**
>
> This list of nodes is automatically filtered depending on the labels the workload has. So this list of nodes might not include all nodes that are registered in the Management System.

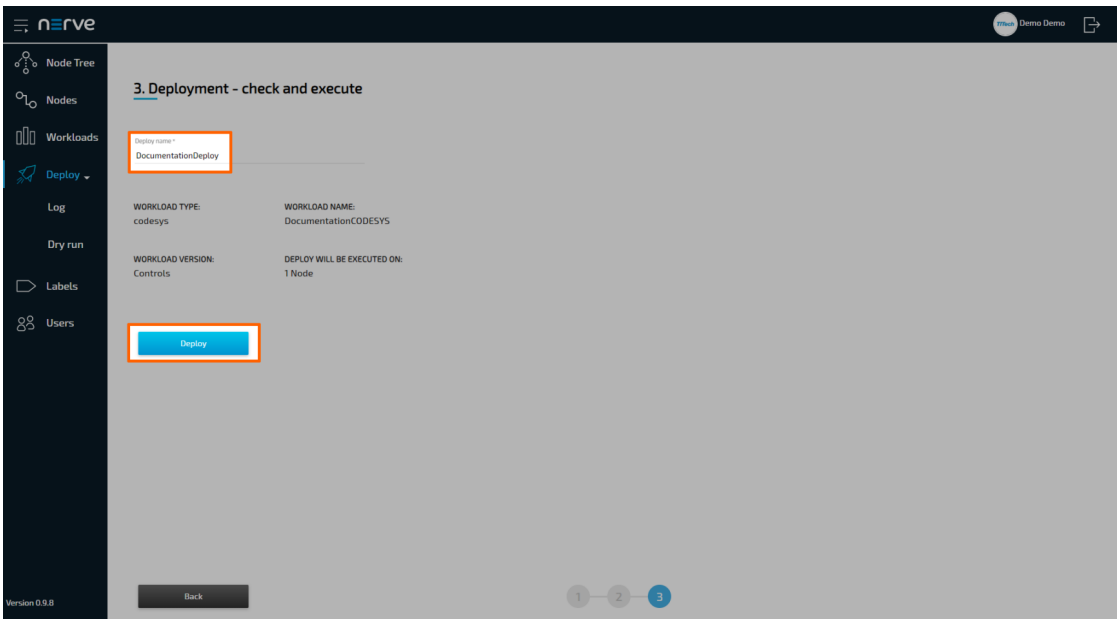
7. Select **Next** in the lower-right corner.



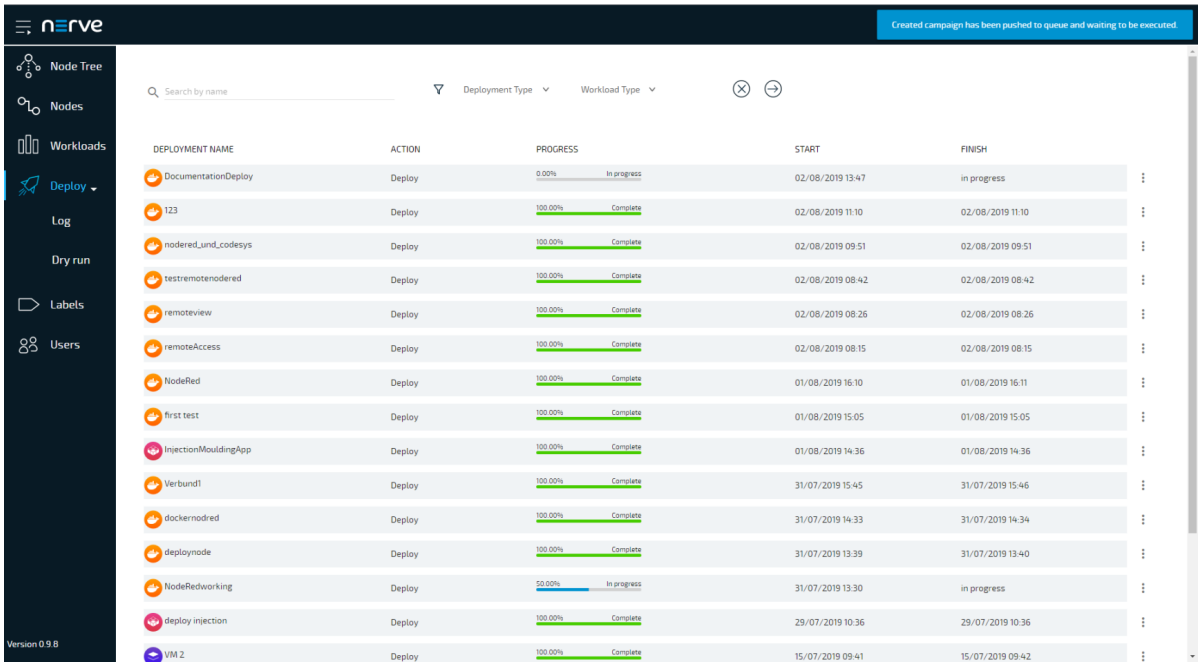
8. In the next window, enter a **Deploy name** above the **Summary** of the workload. Choose a name that makes this deployment easy to identify.

> **NOTE**
>
> You can only use alphanumeric characters (a-z, A-Z, 0-9) and underscore (_). Any other special characters are not allowed. If you use a character that is not allowed, the deploy button will be grayed out.

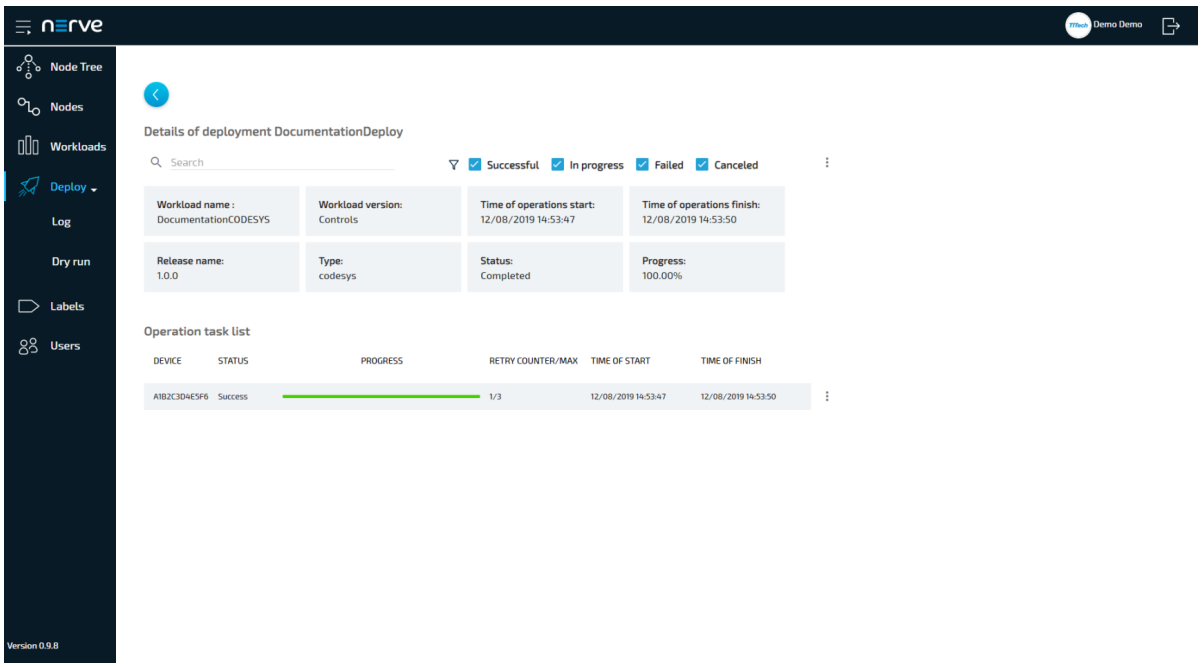
9. Select **Deploy** to execute the deployment.



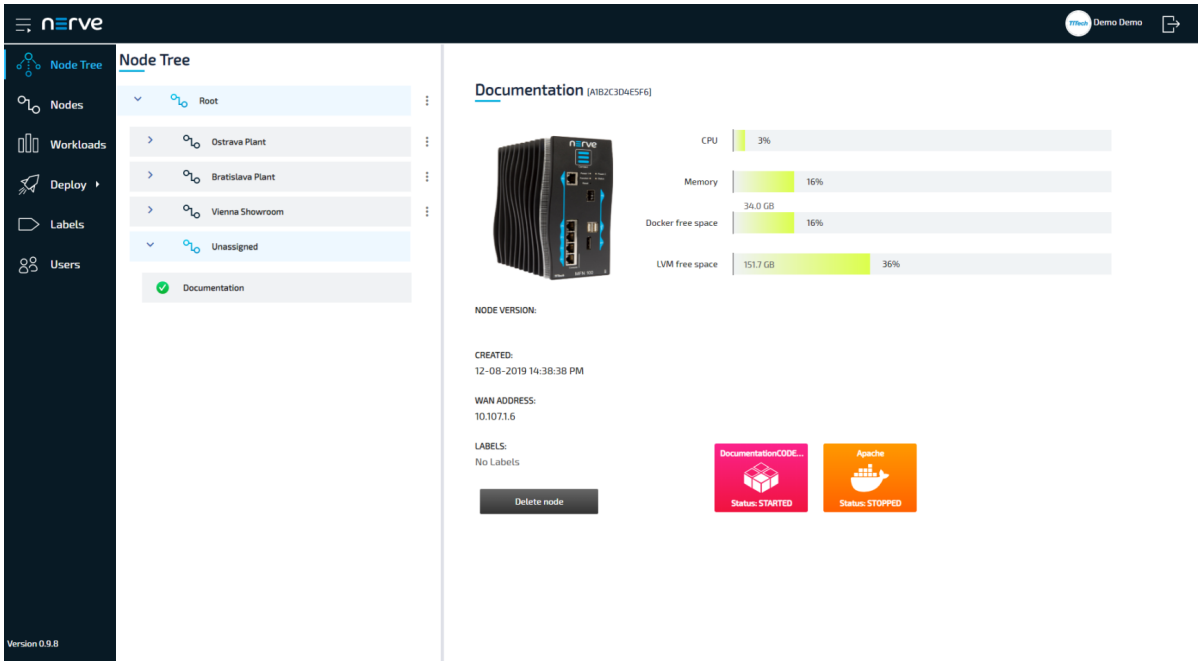
You will be taken to the log next. Your current deployment is at the top of the list. The**Deploy name** you have chosen before is the name that identifies the deployment in the log.



You can see the progress of the current deployment and click the log entry of the deployment to see a more detailed view.



The workload has been deployed and can be controlled in the Node Tree. Select **Node Tree** in the menu on the left and select the node you have deployed a workload to.



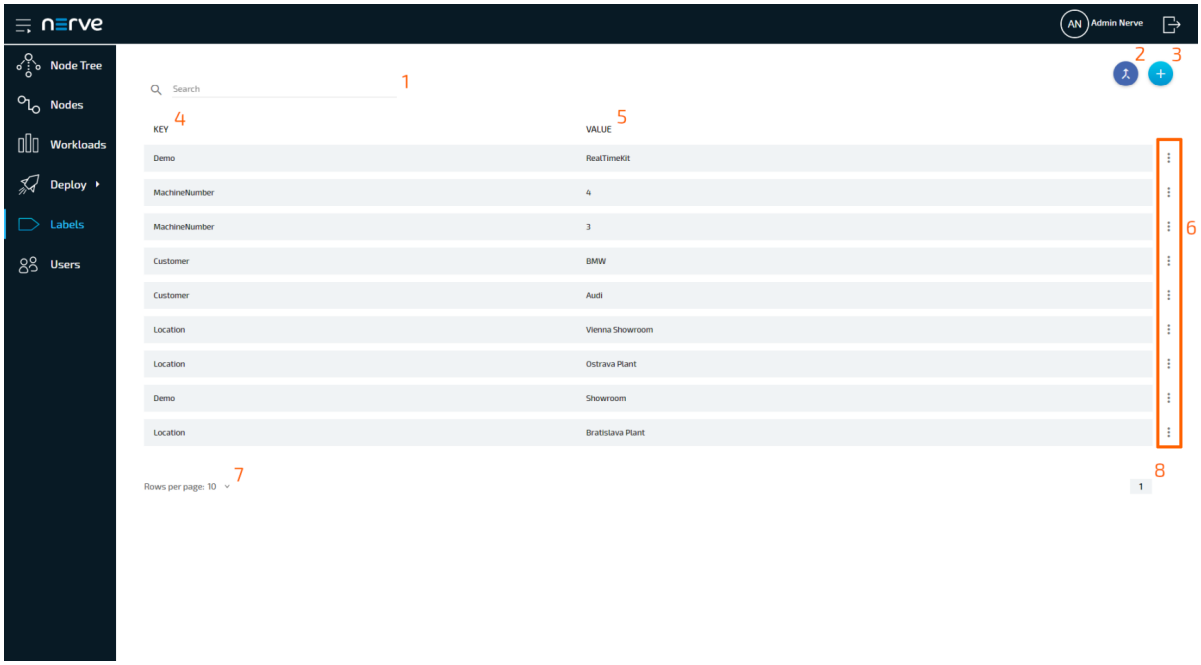
You can control the workloads by clicking the tile of each workload. All workloads are started as soon as they are deployed.

NOTE

CODESYS applications can only be controlled through the local UI.

Labels

Labels are a useful feature that help with the organization of nodes and workloads. They can be defined and used freely. All labels that have been created are listed in the labels menu.

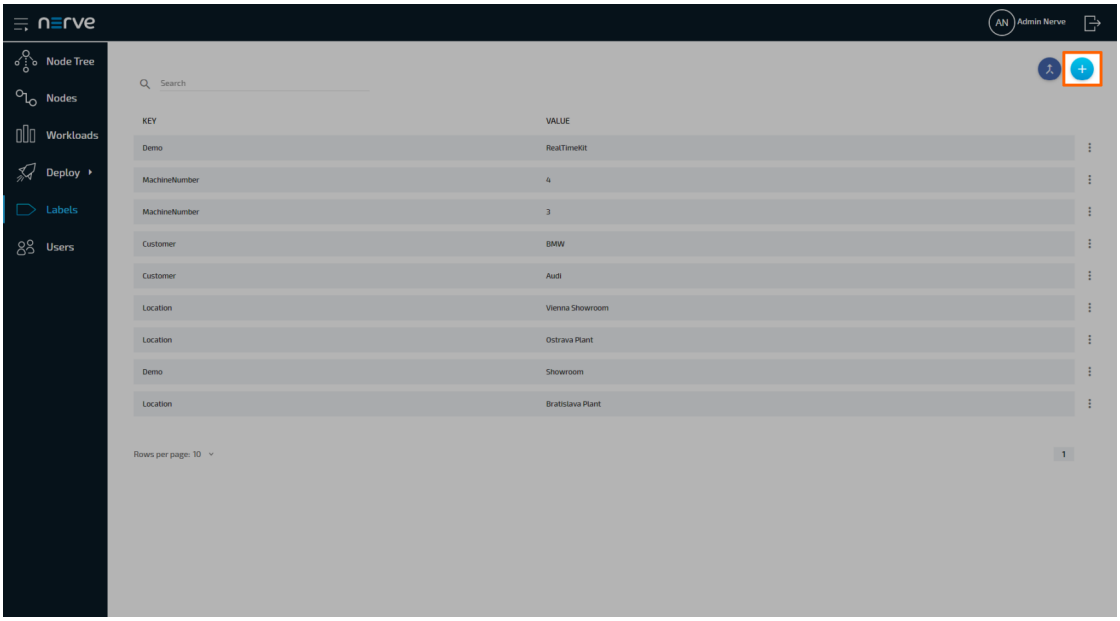


Item	Description
Search bar (1)	Use the search bar to filter labels by key.
Merge labels (2)	Clicking here leads to a page that allows you to merge existing labels.
Add new label (3)	Click here to add a new label consisting of label key and label value.
KEY (4)	This is the key of the label. It can be understood as the "category" of the label. Examples of label keys are location, machine number or hardware.
VALUE (5)	This is the value of the label. It corresponds to the key of the label. Examples are Vienna, Machine 1 or MFN 100.
Ellipsis menu (6)	Clicking here opens an overlay that allows the deletion of labels.
Rows per page (7)	Specify how many labels are displayed on one page. You can select **5**, **10** or **15** labels per page.
Page navigation (8)	Use the arrows to switch between pages. Clicking the number in the middle opens a list of all page numbers. Selecting a number jumps to that page.

Adding a New Label

Labels can be defined with any key and value. They can also be defined in the registration process of a node. When a label is defined there, it is automatically assigned to the node that has been registered.

1. Select **Labels** in the menu on the left side.
2. Click the **Add new label** icon in the upper-right corner.



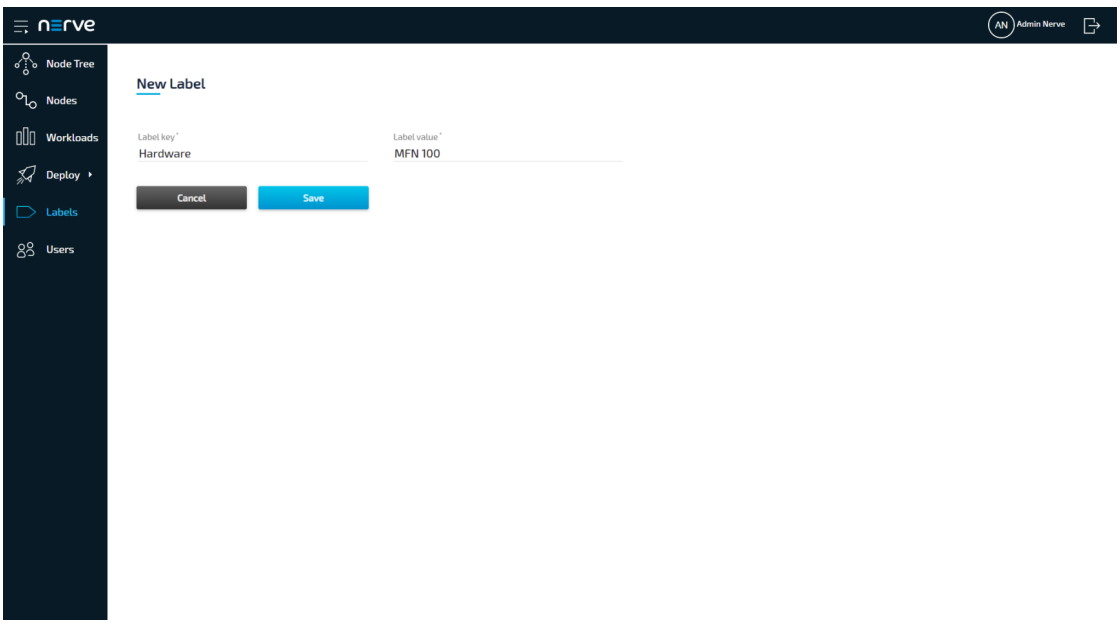
3. Enter the following information:

Item	Description
Label key	Enter the "category" of the label. Example: **Hardware**.
Label value	Enter the value for the label here. Example: **MFN 100**.

NOTE

Label keys must consist of one word only. Please use - and _ as separators.

4. Click **Save** to add the new label.



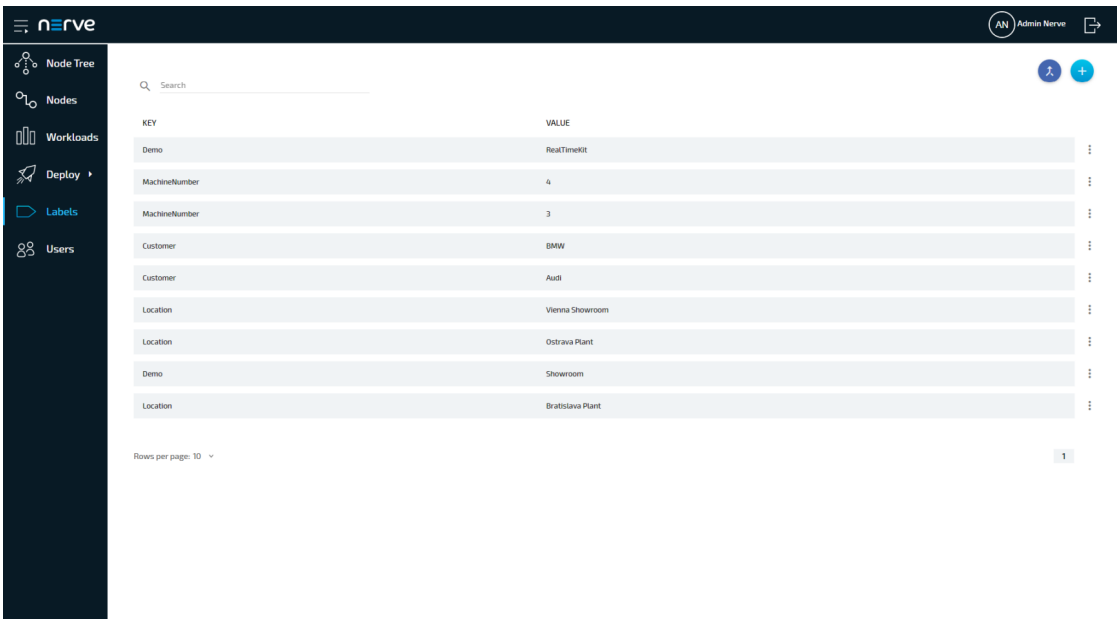
The label now appears in the label list and can be used in the Management System. Labels can be assigned to nodes when node details are edited. In the workload provisioning process they can be chosen as selectors.

Editing a Label

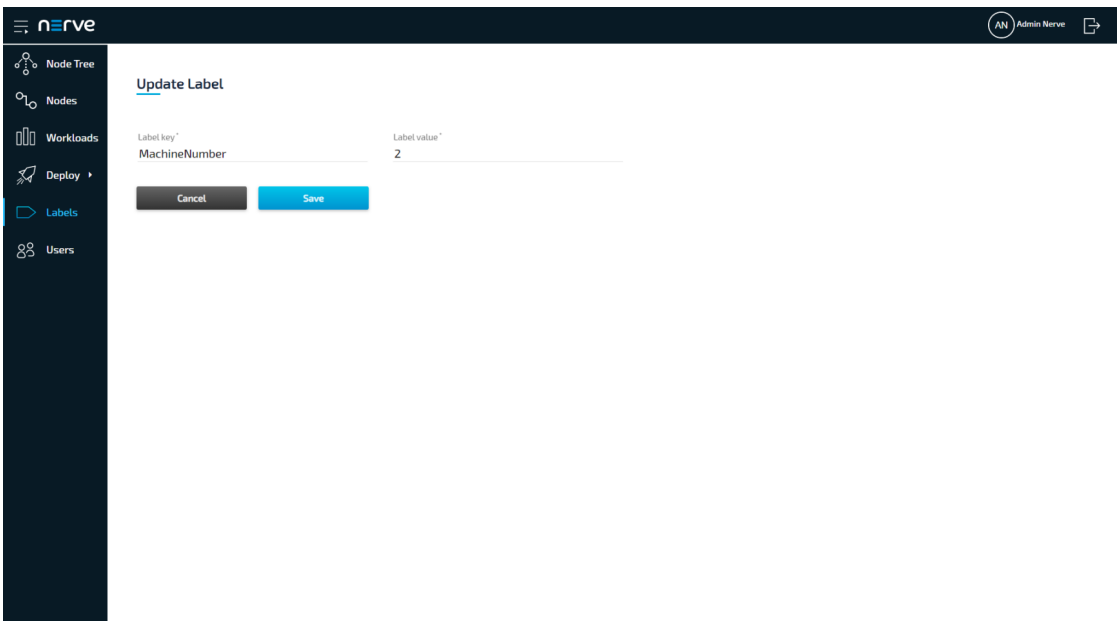
The process of editing a label is very straightforward and virtually identical to the process of creating a new label. Please note that labels will be edited even if they are currently assigned to nodes or used as selectors by workloads.

1. Select **Labels** in the menu on the left side.

2. Click the label you would like to edit.



3. Edit **Label key** and **Label value**.



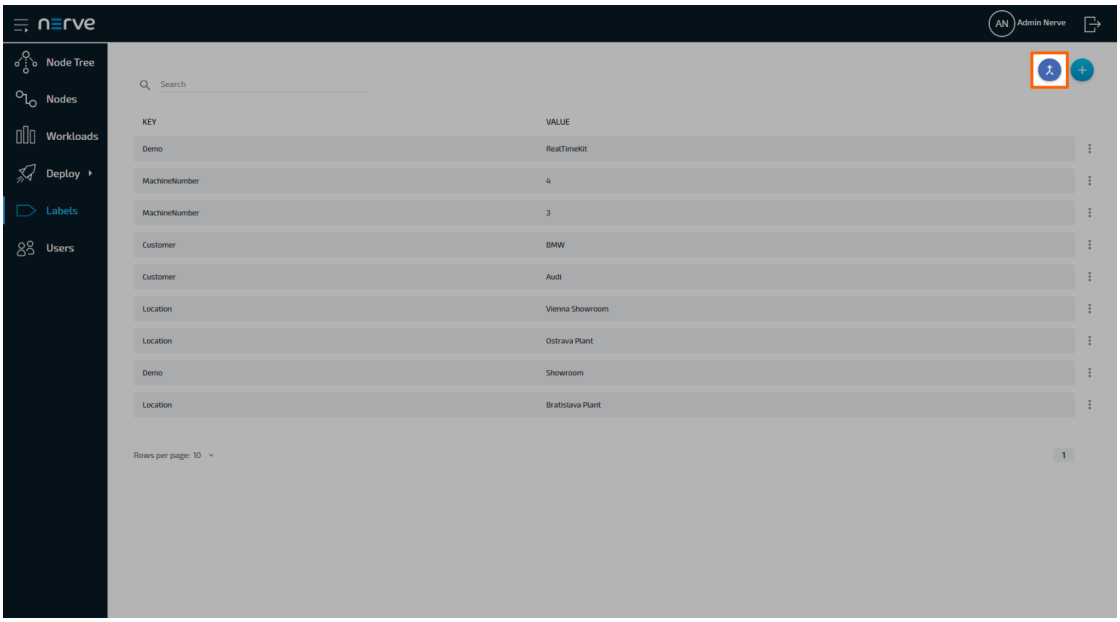
4. Click **Save** to update the label.

Merging Labels

Labels with overlapping information or typos can be merged into one. However, only label keys are merged. The label values are left untouched and assigned to the new label key.

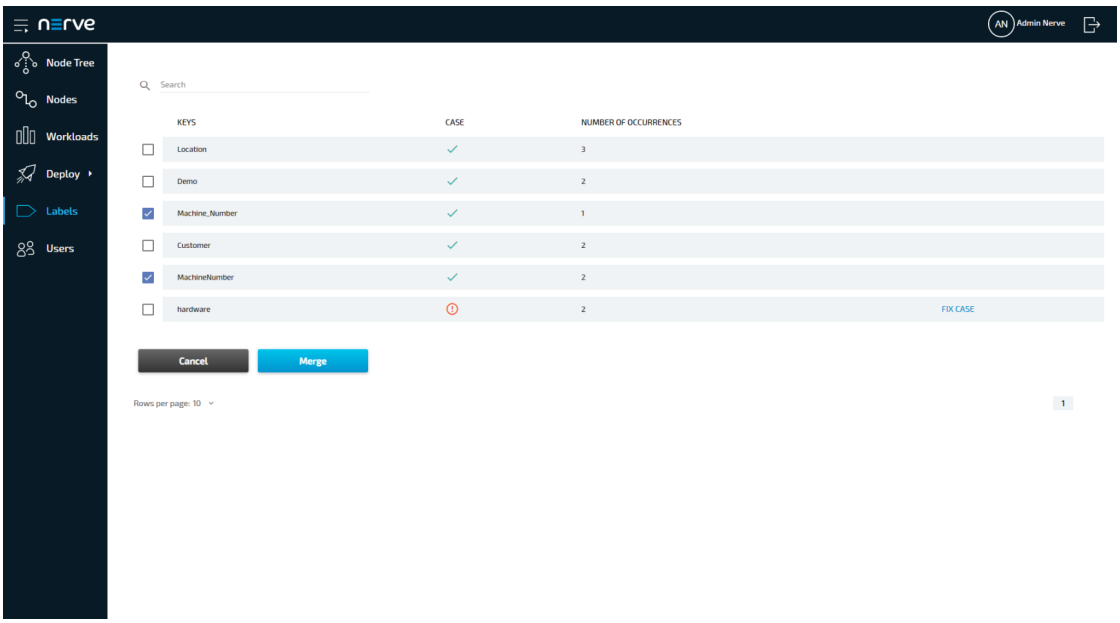
1. Select **Labels** in the menu on the left side.

2. Click the **Merge labels** icon in the upper-right corner.



3. Tick the checkboxes left of the labels you would like to merge. You can select two labels or more.

4. Select **Merge**.



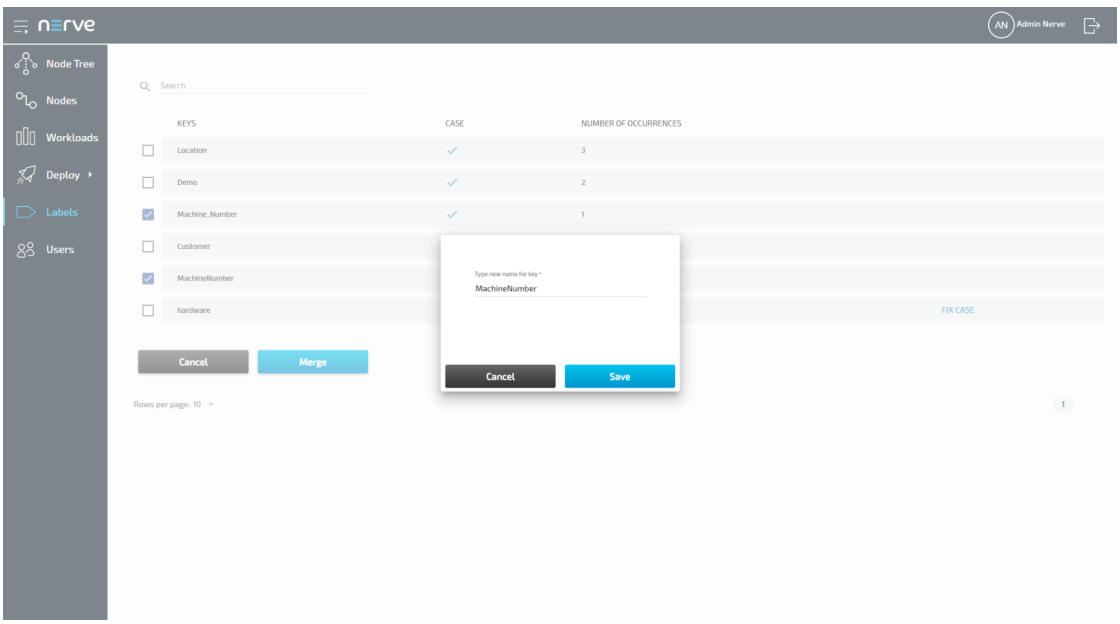
NOTE

The table here gives the following information:

CASE
The system checks if labels that use the same characters are also written in the same case. A green check mark indicates that the case matches. A circled red exclamation mark appears if the case does not match.

NUMBER OF OCCURRENCES
The number here indicates how many labels have been defined with the same label key.

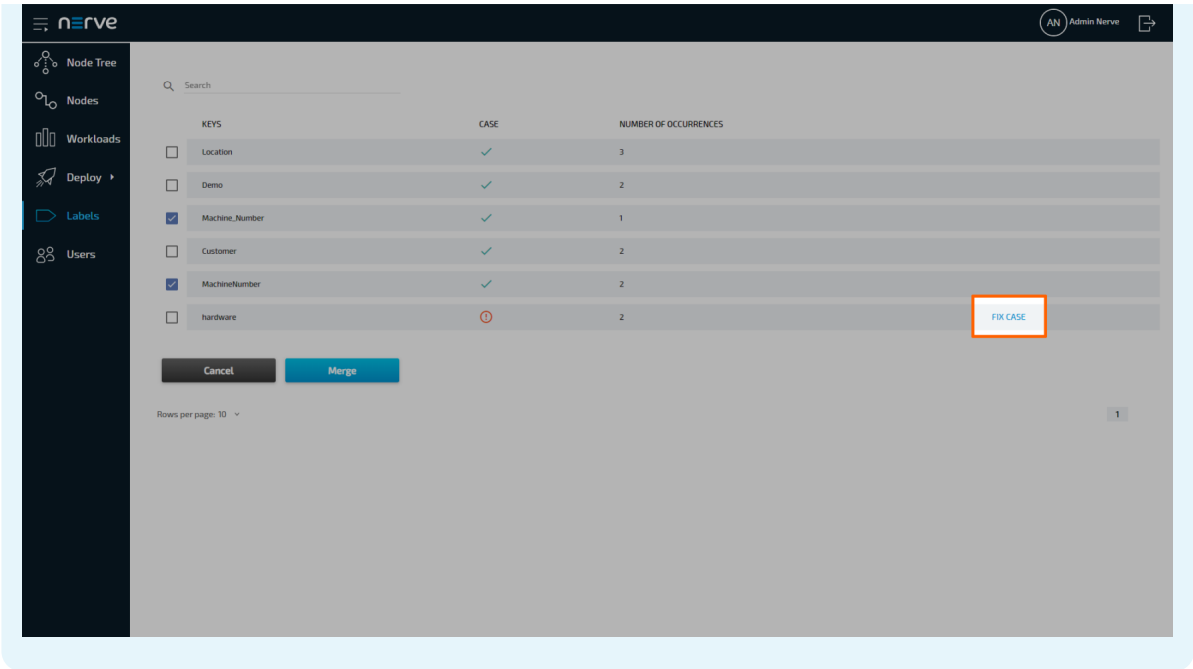
5. Type in the new name for the label key.



6. Click **Save** to save the new label.

NOTE

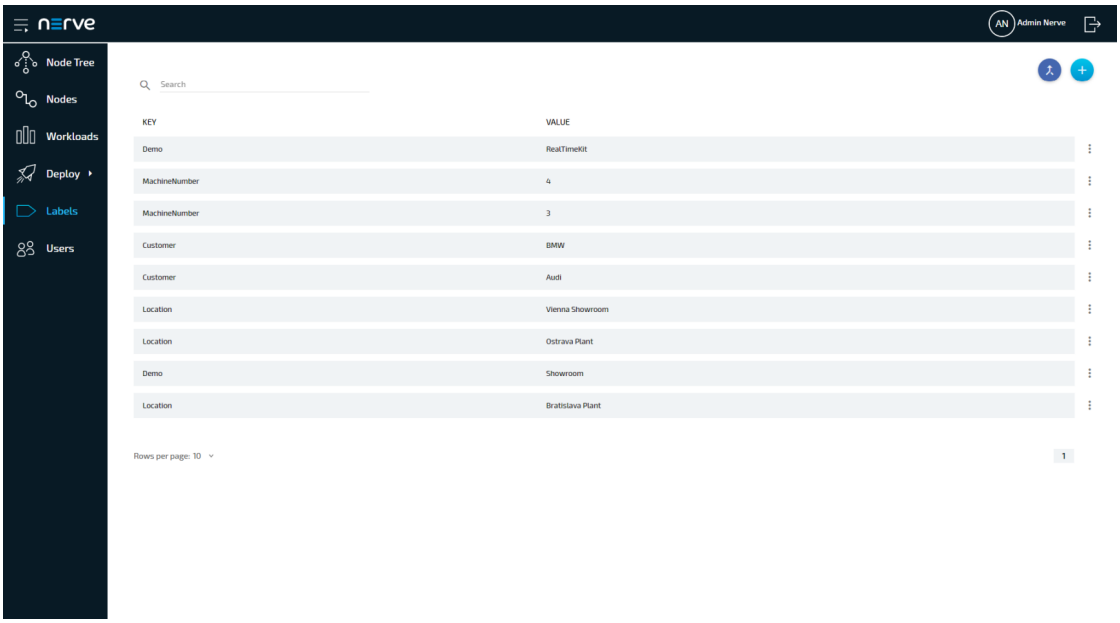
If labels have the same name but are not written in the same case, the Management System will recognize it. In this case, click **FIX CASE** in the list, type in the new name for the label key and select **Save**.



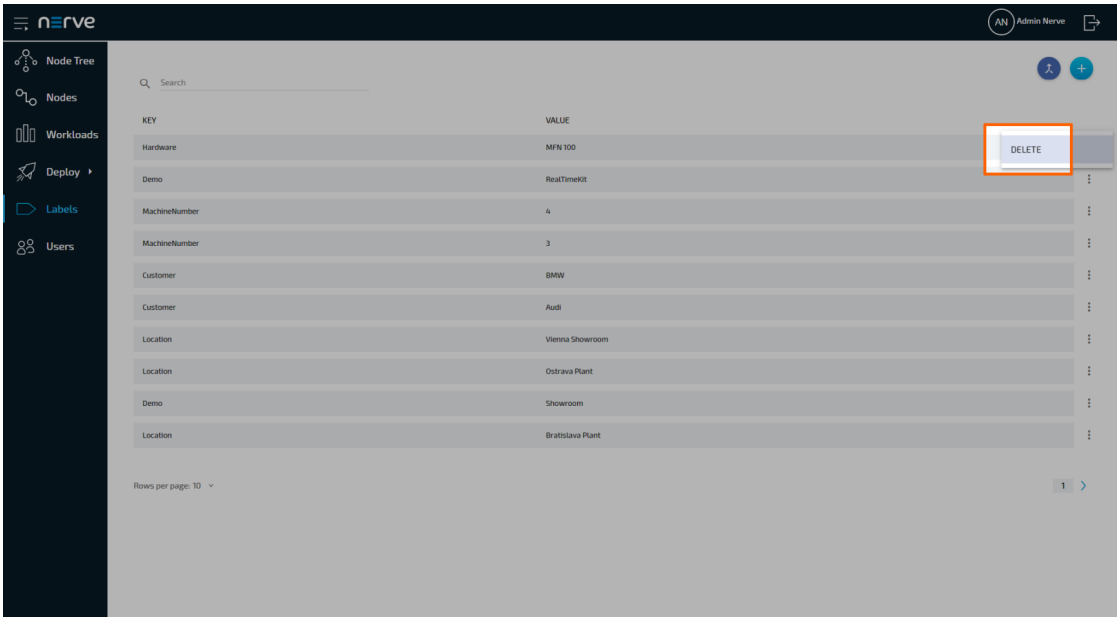
Deleting a Label

Please note that when a label is deleted from the list, it will also be deleted from any node or workload that had the label assigned.

1. Select **Labels** in the menu on the left side.
2. Choose the label you would like to delete.



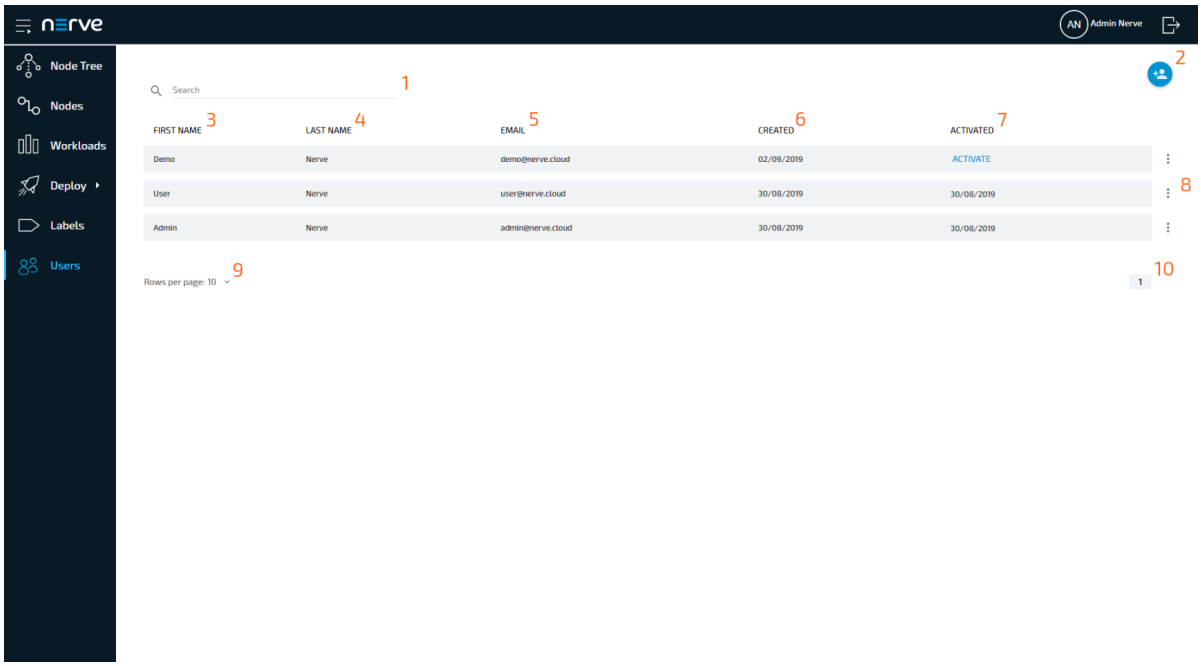
3. Click the ellipsis menu on the right side of the label.
4. Select **Delete** in the overlay that appeared.



5. Click **OK** to delete the label.

User Management

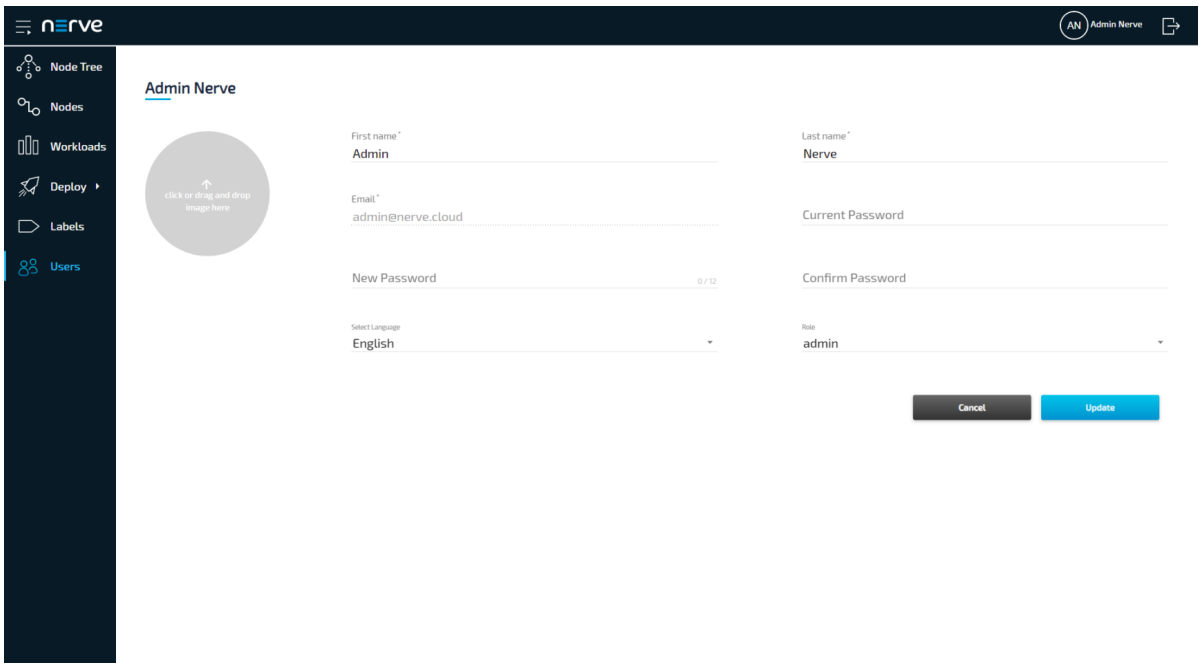
Here you will find a list of all registered users. Every user has their own user profile with details about the user. The default user is admin@nerve.cloud. This user cannot be modified or deleted.



Item	Description
Search bar (1)	Use the search bar to filter the list of users. The columns **FIRST NAME**, **LAST NAME** and **EMAIL** are the targets of the search.
Add new user (2)	Select this icon to add a new user.

Item	Description
FIRST NAME (3)	This is the first name of the registered user. It is displayed in the upper-right corner when the user is logged in.
LAST NAME (4)	This is the last name of the registered user. It is displayed in the upper-right corner when the user is logged in.
EMAIL (5)	This is the e-mail address of the user. It is used as the username for logging in to the Management System. The system also uses this e-mail for sending the activation link and the instructions on how to reset the login password.
CREATED (6)	This is the date the user was created. The date format is DD/MM/YYYY.
ACTIVATED (7)	This is the date the user was activated. The date format is DD/MM/YYYY.
Ellipsis menu (8)	Clicking here opens an overlay that gives two options: **DELETE** and **ASSIGN**.
Rows per page (9)	Specify how many accounts are displayed on one page. You can select **5**, **10** or **15** accounts per page.
Page navigation (10)	Use the arrows to switch between pages. Clicking the number in the middle opens a list of all page numbers. Selecting a number jumps to that page.

Clicking any of the users leads to their user profile. As an admin you can edit the details of any user profile. This includes **First name**, **Last name**, **Select language** and **Role**. The e-mail address of a user cannot be changed. The password for an account can only be changed by the respective user.



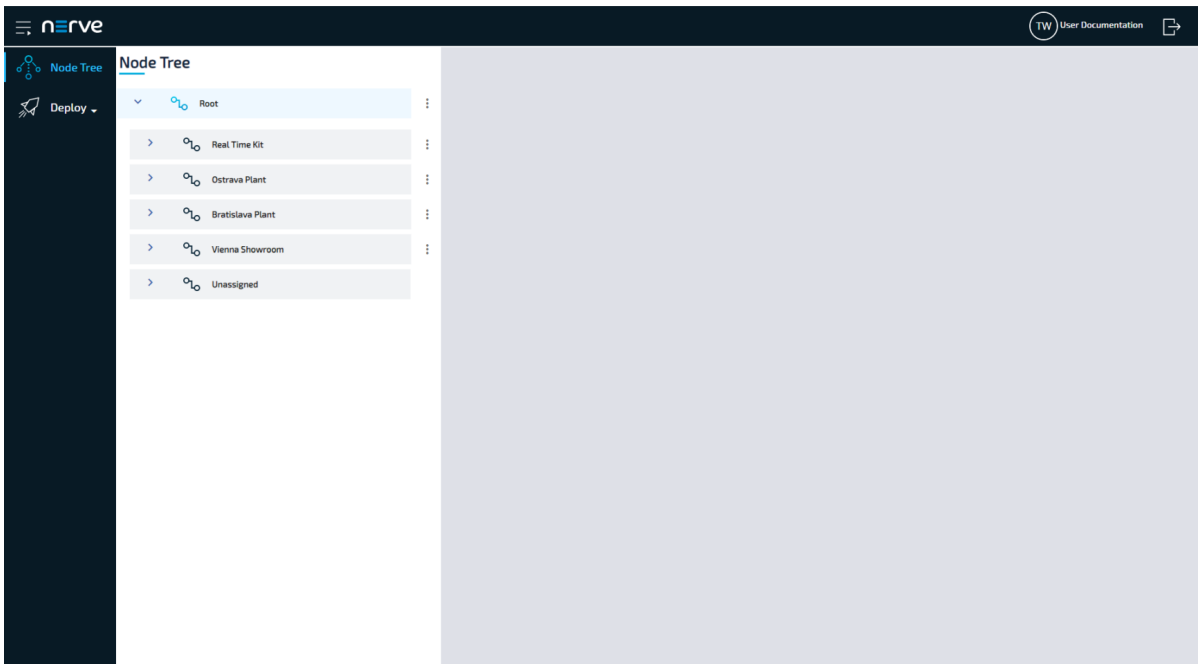
Item	Description
Profile Picture	Click here or drag and drop the image into the gray circle to upload a profile picture for the user. It is displayed in the upper-right corner when the user is logged in.

Item	Description
FIRST NAME	This is the first name of the registered user. It is displayed in the upper-right corner when the user is logged in.
LAST NAME	This is the last name of the registered user. It is displayed in the upper-right corner when the user is logged in.
EMAIL	This is the e-mail address of the user. It is used as the username for logging in to the Management System. The system also uses this e-mail for sending the activation link and the instructions on how to reset the login password.
Current Password	Enter the current password here if you are the active user and want to change your password.
New Password	Enter the new password here if you are the active user and want to change your password.
Confirm Password	Enter the new password again if you are the active user and want to change your password.
Select Language	Select the Management System language from the drop-down menu. English is the only available language at the moment.
Role	This designates the role of the user. Registered users can be **admin** or **users**.

User Roles

In the Management System, there are two user roles: **user** and **admin**. Only users that are assigned an admin role can access all functionality. Therefore, the available features depend on your role. As a rule of thumb, regular users only have read access to the system while admins can modify it.

In the Management System, regular users can only access the **Node Tree** and **Deploy** menus. They can also customize their own user profile.

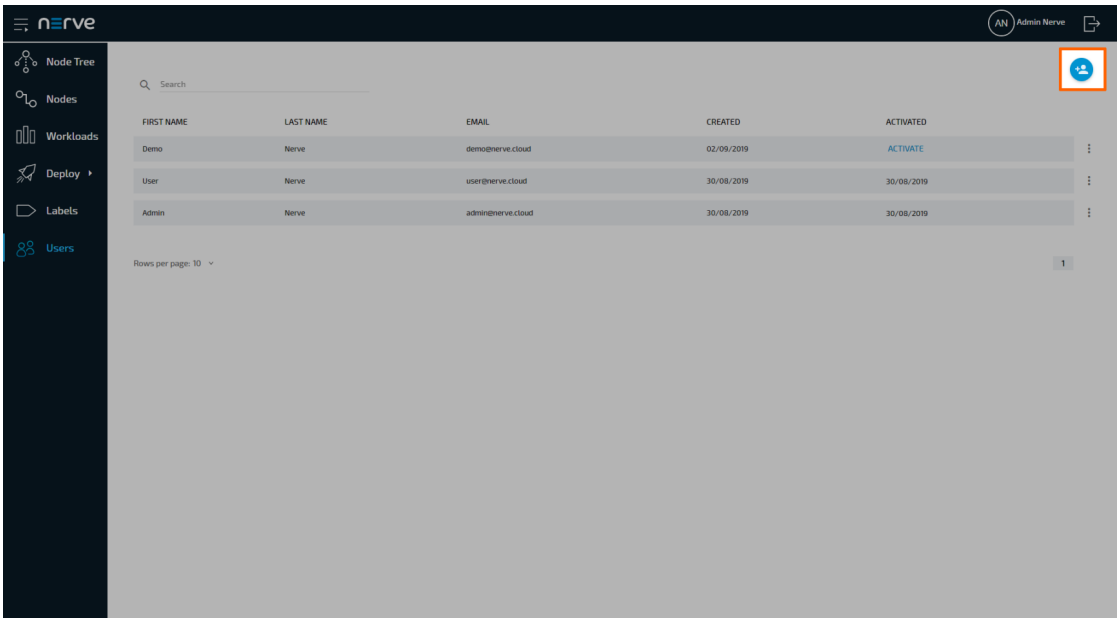


Adding a New User

Users in the admin role have to create new users before these can set a password and log in to the Management System.

1. Select **Users** in the menu on the left side.

2. Click the **Add new user** symbol in the upper-right corner.



3. Enter the required information: **First name**, **Last name** and **Email**.

> **NOTE**
>
> Filling in the remaining fields and setting a profile picture are optional.

4. Select **Save** to create the user.

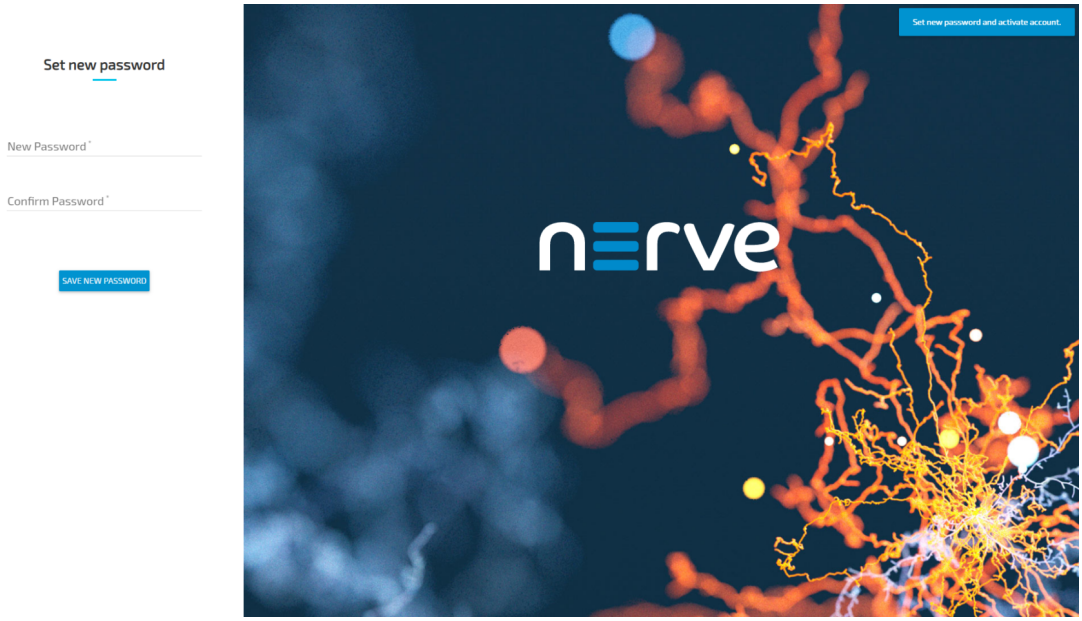
The user needs to be activated after it has been created. This can be done by clicking an activation link that is sent to the e-mail address that was specified during the creation of the user. The activation e-mail is sent automatically by the Management System shortly after the user is created.

Activating a User

To active a user, the user needs to click an activation link that is sent to the e-mail address that was specified during the creation of the user. By following the link, the user needs to set their password in order to be activated and to be able to log in to the Management System.

1. Follow the activation link that was sent to the specified e-mail address.

2. Enter the password you would like to use for Management System access under **New Password** and **Confirm Password**.



NOTE

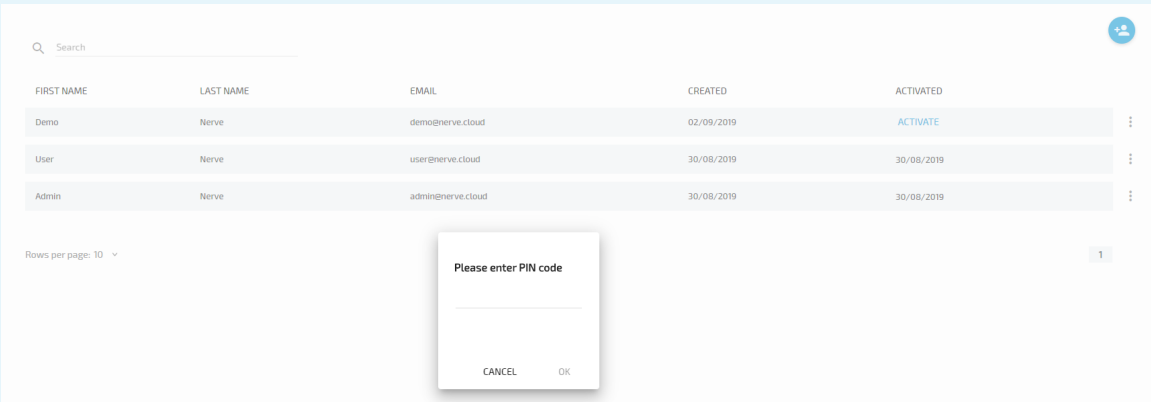
The password must contain at least one uppercase letter, one lowercase letter and one number. It must be at least 7 characters.

3. Select **SAVE NEW PASSWORD**.

The new user is now activated and can log in immediately with their e-mail address and the password they have defined.

NOTE

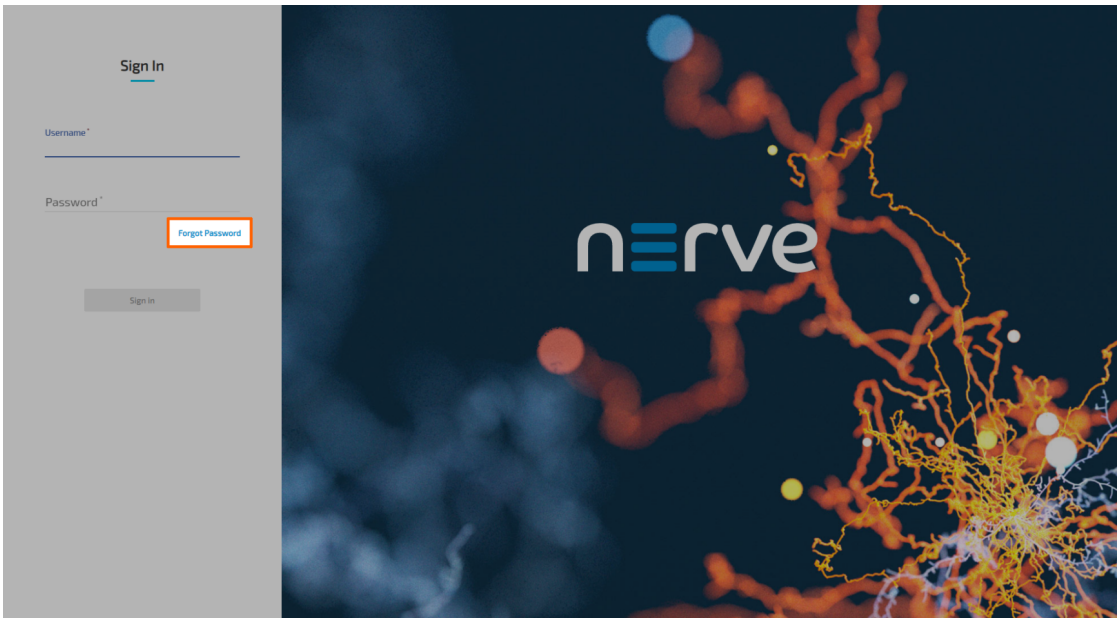
When an admin creates a new user, the Management System states **ACTIVATE** in the **ACTIVATED** column. Clicking this opens up a new window, in which the Management System asks for a PIN Code. This is a relic and has no function. It will be removed in a future update so please ignore this for now.



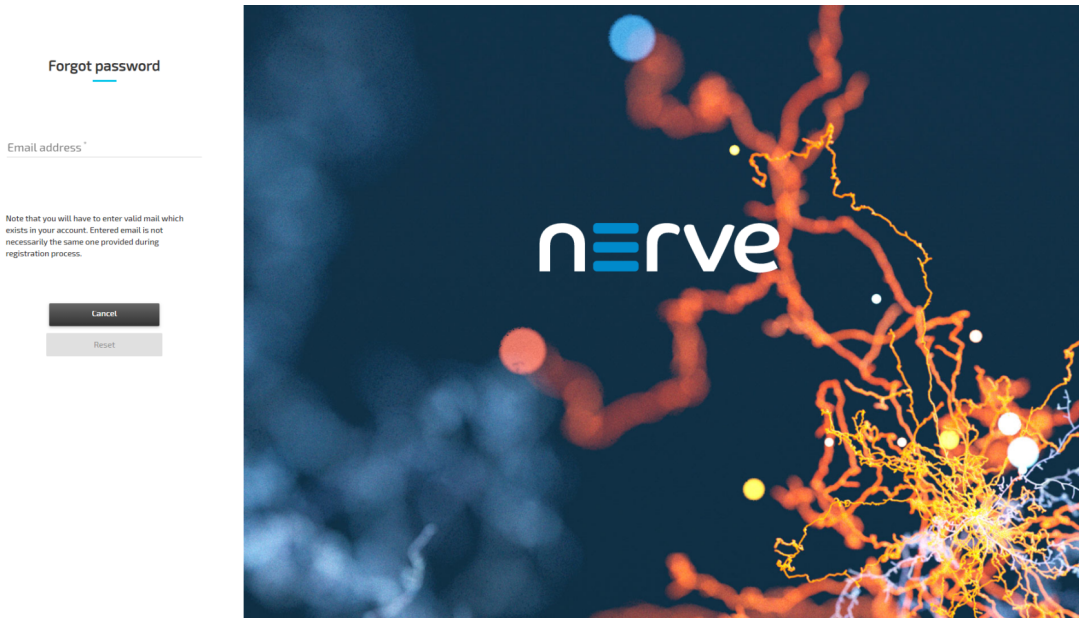
Resetting the Password

In case a user does not remember their password, a request to reset the password can be sent from the login page of the Management System.

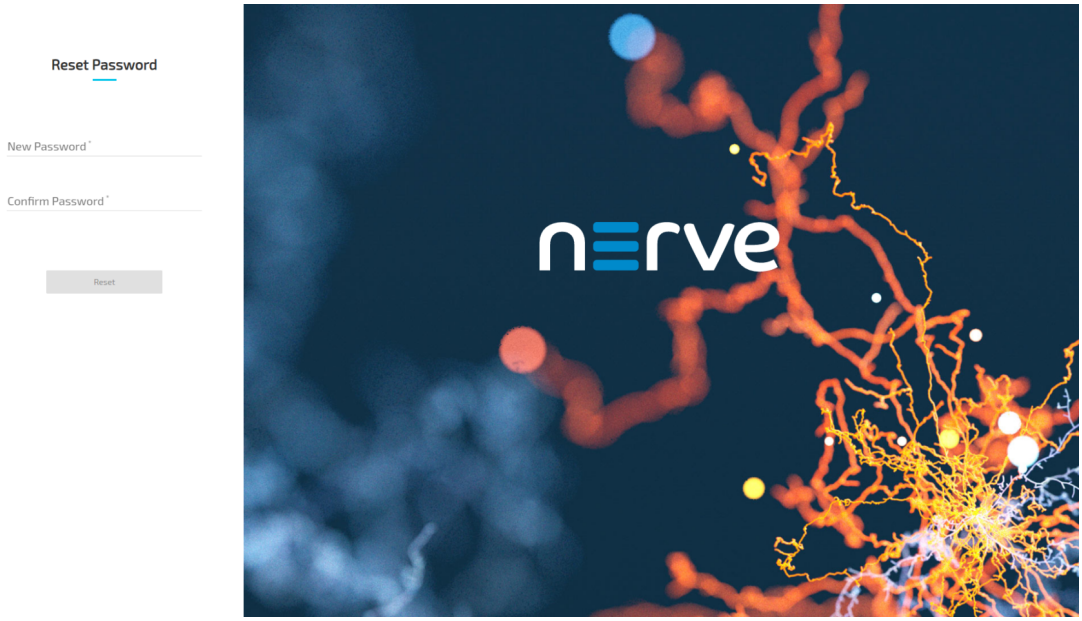
1. Access your Management System.
2. Select **Forgot Password**.



3. Enter the e-mail address of the account that needs to reset the password.



4. Select **Reset**. An e-mail with instructions is sent to the e-mail address of the account.
5. Follow the link from the e-mail to reset the password.
6. Enter the password you would like to use for Management System access under **New Password** and **Confirm Password**.



7. Select **Reset** to save the new password.

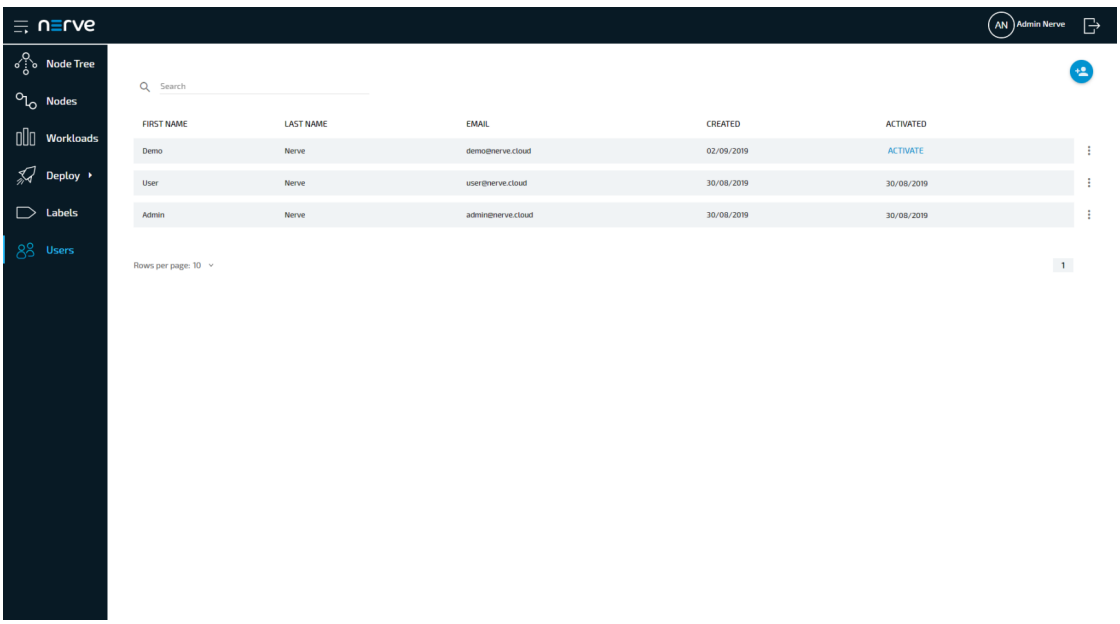
The password has been changed and the user can log in immediately with their e-mail address and the new password.

Deleting a User

Users that are assigned the role of admin can delete any user that is registered in the Management System.

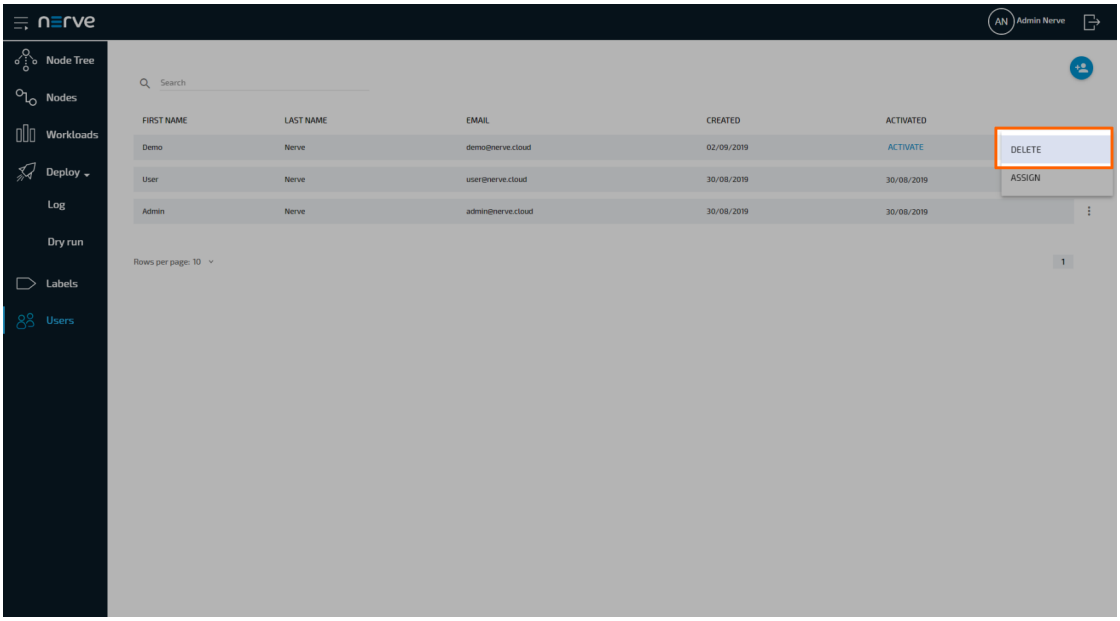
1. Select **Users** in the menu on the left side.

2. Choose the user you would like to delete.



3. Select the ellipsis menu to the right of the user in the list.

4. Select **DELETE** from the overlay that appeared.



5. Click **OK** to confirm the deletion of the user.

NOTE

The ellipsis menu also offers an **Assign** option. This is a leftover of a feature that is now fully automatic and serves no function anymore. This will be removed in the near future so please ignore this for now.

Node Internal Networking

This chapter explains how a user can connect workloads (VMs and Docker containers) to services and network ports of a node. In order to do this, it explains the internal networking concepts in detail. Most workloads will need to be connected to a network as networking is the main form of communication for workloads. They either want to connect to external servers or they are servers themselves, in which case they need to be made visible for their communication partners. The Nerve networking system enables both use cases.

The image below shows an example node consisting of the **host/domain-0** and the real-time VM running the CODESYS runtime (labeled **codesys**). To further clarify the networking example it also has one Virtual Machine workload and two Docker workloads deployed. The virtual machine is located outside of the host and the Docker containers are located in the Docker network inside of the host. For the sake of the example, however, the workloads are not yet connected. This is done in the examples further below.

The physical ports **P1** to **P5** and **I/O** of the Nerve Device (the MFN 100 in this case) are displayed on the left, touching the host. The black interfaces connected to them inside the host are Linux bridged interfaces displayed with their names on the host. Highlighted by a purple dashed frame is the libvirt network with NAT interfaced being on the left and isolated interfaces on the right. Slightly above them is the **rtvm** interface. Highlighted by an orange dashed frame is the Docker network including the Docker gateway (the orange interface labeled **docker0**). All interfaces colored in red are related to the CODESYS runtime. Interfaces labeled **eth** are symbolic

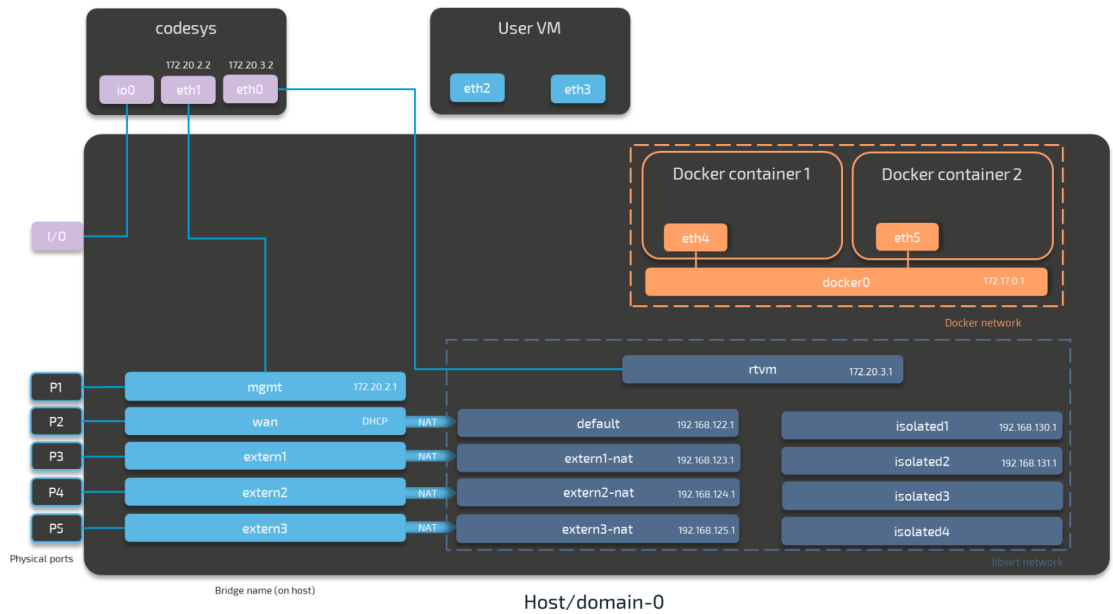
representations of interfaces that are used by virtual machines and Docker containers for communication with the Nerve Blue system. The actual interfaces used depend on the Docker container or virtual machine.

Connections are displayed in three ways. Blue lines are connections that are predefined by the system. Blue arrows are used between bridged interfaces and the libvirt network to indicate NAT. Orange lines indicate the connection between Docker containers and the Docker gateway inside the default Docker network. Further down below, green lines are used as example connections that can be done by the user.

As mentioned above, the image below represents the MFN 100. Please see the device guide for information on your Nerve Device as the physical ports and the connection to their respective interfaces differ.

NOTE

codesys refers to the name of the CODESYS runtime in the Nerve Blue system. When the CODESYS Development System is referenced, CODESYS is always written in capital letters.



Please see the table below for more information on the interfaces, their usage and their IP ranges.

Legend	
	These ports are device dependent. They are included here for clarification of the image above. Please see the device guide for information on your Nerve Device.
Physical Ports	• **P1 to P5** Ethernet ports of the MFN 100. Please note that P5 is an SFP port. • **I/O** While also an Ethernet port, this port is reserved for communication with the CODESYS runtime.

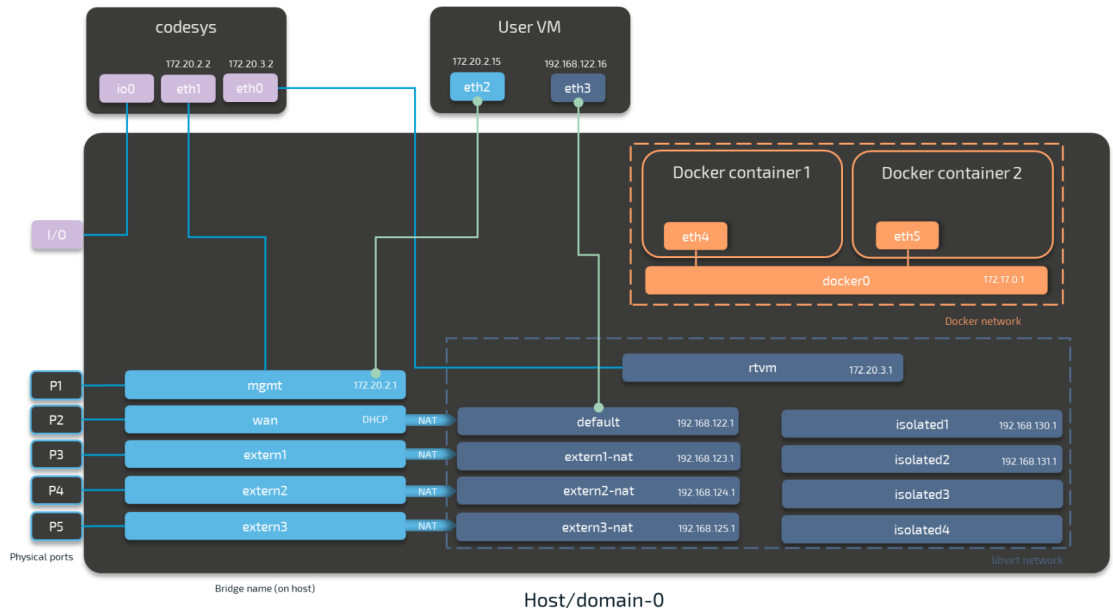
Legend	
Bridged Interfaces	• **mgmt** This interface is mainly used for the configuration of the system. It is connected to the **eth1** interface of the CODESYS runtime. • **wan** This interface is designated for internet connection. • **extern1, extern2, extern3** These interfaces can be used for external connections.
NAT Interfaces	IP addresses are assigned by a DHCP server when NAT interfaces are used. The IP address is assigned in subnet 255.255.255.0. • **default** This network is the NAT interface of the **wan** bridged interface with an IP address in the range from 192.168.122.2 to 192.168.122.254. • **extern1-nat, extern2-nat, extern3-nat** These networks are the NAT interfaces of the **extern1**, **extern2** and **extern3** bridged interfaces. The IP addresses of these interfaces are in the following ranges: **extern1-nat** 192.168.123.2 to 192.168.123.254 **extern2-nat** 192.168.124.2 to 192.168.124.254 **extern3-nat** 192.168.125.2 to 192.168.125.254
Isolated Interfaces	Isolated interfaces can be used to allow communication between two virtual machines. These networks cannot communicate outside of the system. • **isolated1** This interface has a DHCP server, meaning that a VM can configure a static or dynamic IP address in the range from 192.168.130.2 to 192.168.130.254. • **isolated2** This interface has a DHCP server, meaning that a VM can configure a static or dynamic IP address in the range from 192.168.131.2 to 192.168.131.254. • **isolated3** This network does not have a DHCP server. • **isolated4** This network does not have a DHCP server.

Legend	
	Other interfaces that can be used as a NAT network but without port forwarding. These interfaces do not communicate outside of the system.
Other interfaces	• **rtvm** Use this interface for communication with the CODESYS runtime. • **docker0** This is the Docker gateway designated for communication with Docker. Interfaces of Docker containers that are connected to the gateway receive IP addresses in the default Docker network, ranging from 172.17.0.2 to 172.17.0.244. • **eth** These interfaces are symbolic representations of interfaces that are used by virtual machines and Docker containers for communication with the Nerve Blue system. The actual interfaces used depend on the Docker container or virtual machine. The only exceptions are **eth0** and **eth1** as they are always defined with their names for the CODESYS runtime. • **io0** This interface is defined for communication between the I/O port and the CODESYS runtime.
Connections	• **Blue lines** Blue lines signify connections that are predefined by the Nerve Blue system. • **Green lines** Green lines are example connections that could be made by a user to connect virtual machines or Docker containers to the network. • **Orange lines** Orange lines indicate a connection between the interface of a Docker container and the Docker gateway **docker0**. • **Blue arrows** Blue arrows indicate a NAT between bridged interfaces and the libvirt network.

The following sections are conceptual explanations. Workloads are attached to internal networks during the provisioning process. Please refer to the provisioning chapters (Virtual Machine workloads and Docker workloads) in the user guide on how to provision workloads.

Attaching Virtual Machines to a Network

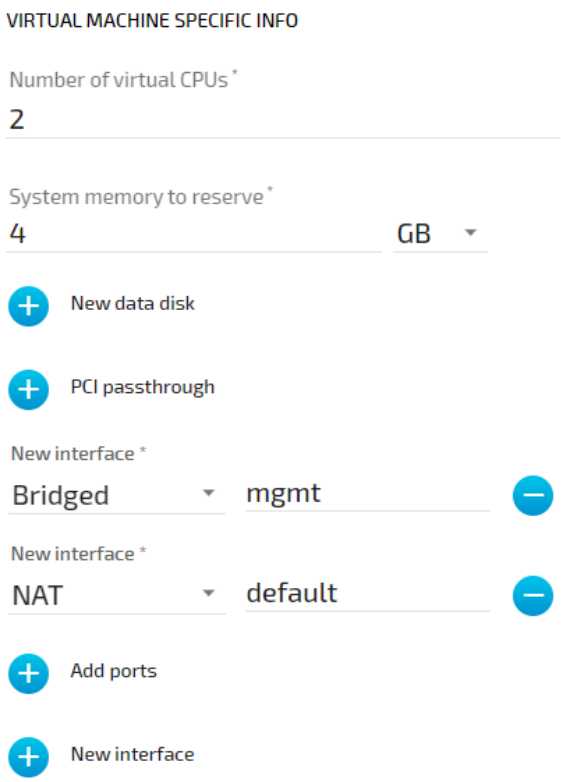
Virtual machine networking is comparable to installing a network card in the virtual machine and attaching it to the network with the network name given in the network drawing. For this example, there are two "network cards" installed in a user deployed virtual machine. They are labeled **eth2** and **eth3** in this example. Green lines indicate a user established connection.



There are two connections established here: **eth2** is connected to the **mgmt** bridged interface for communication with the CODESYS runtime inside of the system. **eth3** is connected to the **default** NAT interface for an internet connection protected by NAT on **P2** of the Nerve Device. Both interfaces have IP addresses in the designated ranges. 172.20.2.15 for **eth2** was manually configured in the virtual machine and 192.168.122.16 for **eth3** was assigned by the DHCP server.

Settings Example

To achieve the functionality above, configure the interfaces of the Virtual Machine workload the following way during the provisioning process in the Management System:

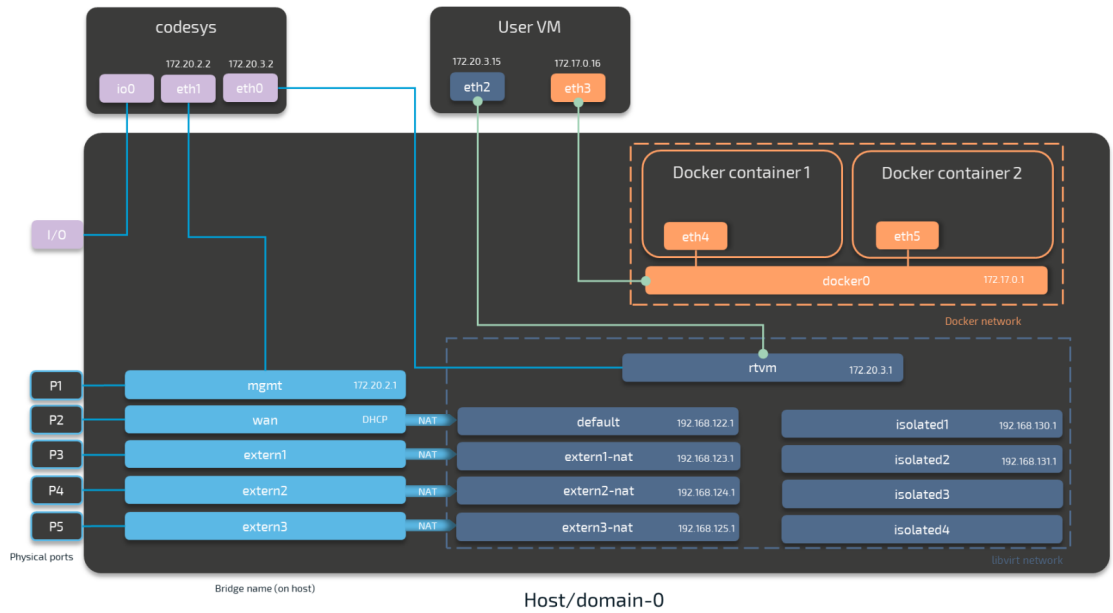


Communication of a Virtual Machine with a Docker Container and the CODESYS Runtime

For Docker containers the situation is different. Docker containers are automatically attached to the Docker default network and access other parts of the system through this network. It is typically called **docker0** and has the IP address 172.17.0.1 assigned. In order to make a server accessible for other workloads, you need to map the port and protocol of the Docker container to the outside by specifying this during workload provisioning in the port mapping section.

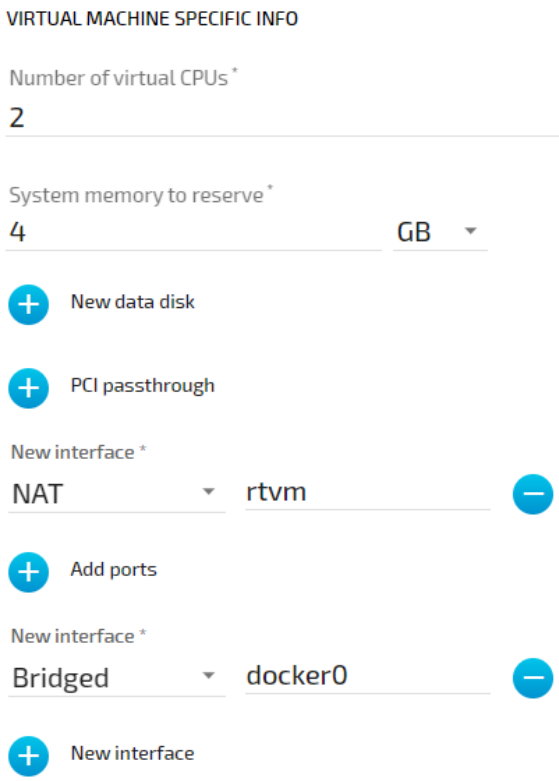
A virtual machine can communicate with Docker containers by connecting an interface to the Docker gateway **docker0**. In the example below this is done with interface **eth3** that has the IP address 172.17.0.16 in the default Docker network. The IP address was manually configured.

Workloads can also be attached to the CODESYS runtime through Nerve networking. For this, the "network card" of the virtual machine is attached to the **rtvm** network.



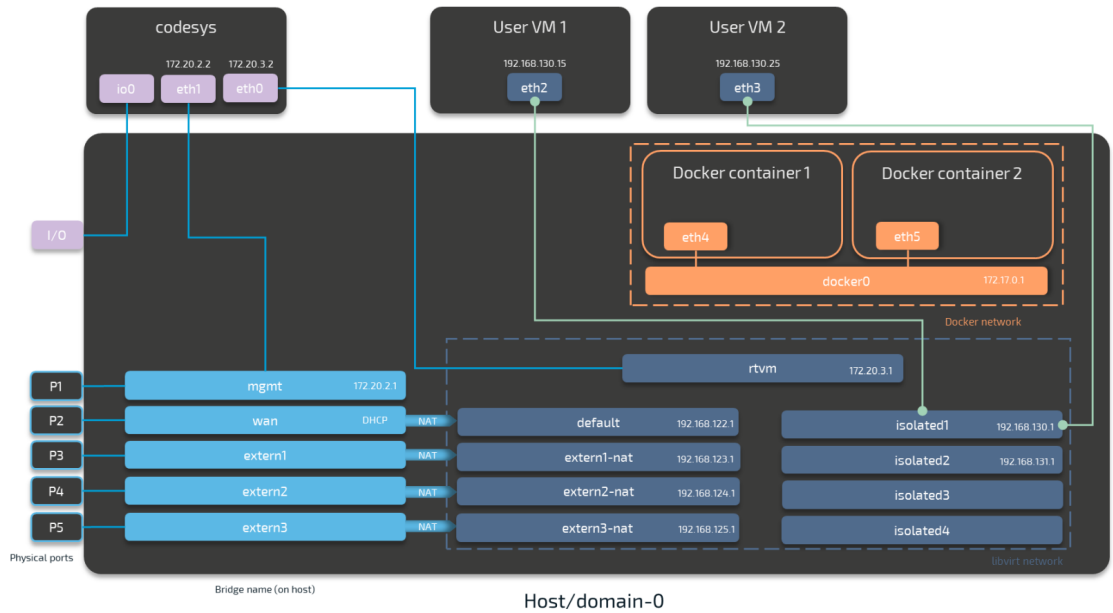
Settings Example

To achieve the functionality above, configure the interfaces of the Virtual Machine workload the following way during the provisioning process in the Management System:



Communication of Two Virtual Machines through Isolated Networks

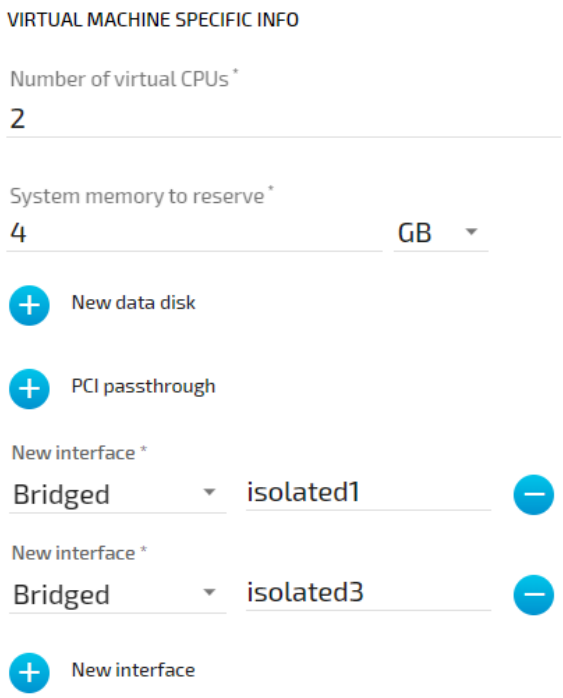
Nerve Blue offers isolated network interfaces for communication of workloads inside of the system. These interfaces do not communicate outside of the system. They can be used to establish communication between two Virtual Machines.



Both virtual machines have a "network card" installed. **User VM 1** is connected to the **isolated1** interface through **eth2** and **User VM 2** is connected through its interface **eth3** to the same network interface, **isolated1**. Each interface has been assigned an IP address by a DHCP server in the designated range: 192.168.130.15 for **eth2** and 192.168.130.25 for **eth3**.

Settings Example

To achieve the functionality above, configure the interfaces of the Virtual Machine workload the following way during the provisioning process in the Management System:



List of Reserved TCP/UDP Ports

The following list states ports that are reserved in version 2.0.

Port	Interface	Protocol	Reserved for
22	none	TCP	SSH daemon
514	127.0.0.1	TCP/UDP	System Log
1883	127.0.0.1	TCP	Local MQTT broker
1884	127.0.0.1	TCP	Local MQTT broker
3000	172.20.2.1	TCP	Local UI
9000	127.0.0.1	UDP	Filebeat

Device Guide

Device Guide

The device guide is an extension of the user guide. It gives an overview of supported Nerve Devices and the device specific information that is required for operating Nerve Blue software.

Each chapter of a device includes the following information:

- Technical Data
- Hardware Overview and Setup
- Device Specific Configuration

- Feature List (for third party devices)
- Network Schematic

NOTE

The first available Nerve Device is the MFN 100. Third party devices are being certified for Nerve Blue usage. The device guide will be updated with information about these devices as soon as they are ready.

MFN 100

The MFN 100 is a qualified Nerve Device that is optimized and tested for use with Nerve software. The device is designed for use in harsh industrial environments (-40 °C to +70 °C). It is based on an Intel Atom x5-E3940/50 CPU and offers 4 GB/8 GB RAM and up to 512 GB SSD storage. The MFN 100 offers one I/O port for Ethernet-based fieldbus connectivity, four GbE switch ports and one SFP port. Additional interfaces include two USB 2.0 ports and one Display Port.

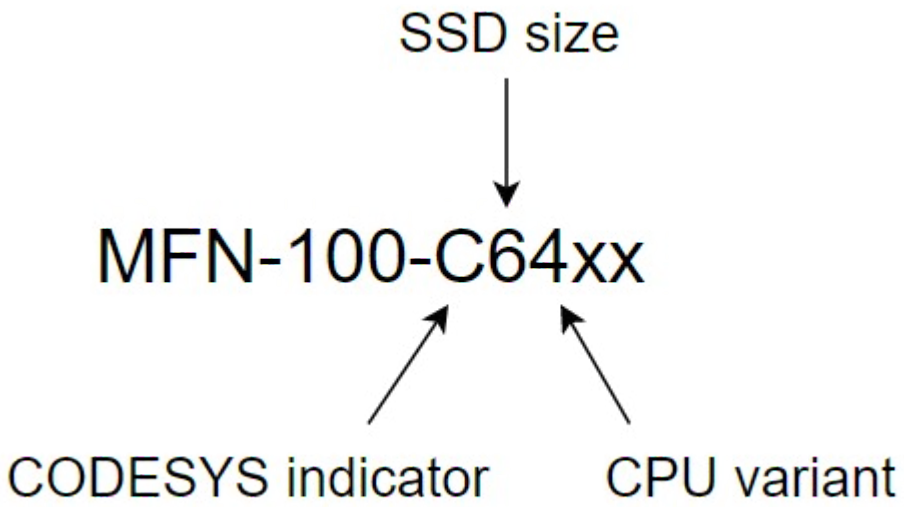
Technical Data

CPU	**Intel E3940** 4 cores, 1.8 GHz, 4 GB RAM **Intel E3950** 4 cores, 2.0 GHz, 8 GB RAM
Storage	64 GB SSD MLC 256 GB SSD MLC 512 GB SSD MLC
Performance	1 ms control cycle time achievable with Nerve Blue
Interfaces	• 4 x< RJ 45 Ethernet (1000/100/10 Mbit/s) • 1 x SFP (1000 Mbit/s) Optical transceivers / OFCS modules may be used which are in compliance with Class I device acc. 21 CFR 1040 and IEC/EN 60825-1 • 1 x DP++ • 2 x USB 2.0 1 A combined current
Mounting	DIN rail or wall mount
Dimensions	(h x w x d): 179 x 87 x 143 mm
Weight	2.1 kg
Power	2 x 24 V redundant input, Average power consumption 12 W
Environmental Parameters	• Operating Temperature Range: -40 °C to 70 °C • Shock and Vibration: ISO 60068-2-27: 15 g peak, 11 ms ISO 60068-2-6: 5 Hz <= f < 8.4 Hz: 3.5 mm, 8.4 Hz <= f <= 150 Hz: 1.0 g • IP 40 according to IEC 60529 • Indoor use only, intended for use in control cabinets • Use up to pollution degree 3 • Use only in environments where no condensation will occur • Maximum relative humidity: 80% for temperatures up to 31 °C decreasing linearly to 50% relative humidity at 40 °C • Maximum altitude: 2000 msl
Certificates	CE and UL certified (EN 61000-6-2/4, IEC/UL 61010, CSA C22.2 NO. 61010-1-12)

Hardware Overview and Setup

Identifying the MFN 100

The label of the MFN 100 can be found on the back of the device, close to the DIN rail clip. Exact identification is possible through the combination of product number (P/N), serial number (S/N) and version number (V/N) that are printed on the label. The model number of the MFN 100 details the variant of the MFN 100 you have purchased:



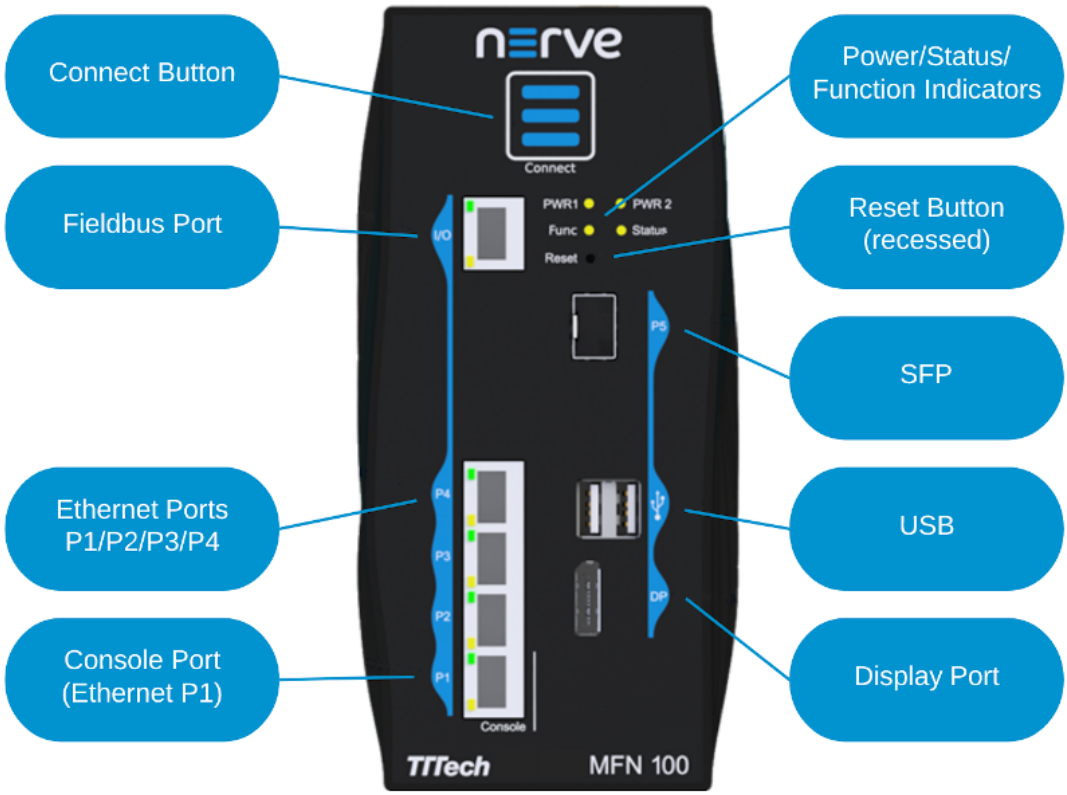
Letter or Number	Description
CODESYS indicator	This letter indicates whether the device has a CODESYS runtime pre-configured: • **C** — The CODESYS runtime is pre-configured • **X** — The CODESYS runtime is not pre-configured
SSD size	This number indicates the size of the SSD: • **6** — 64 GB SSD • **2** — 256 GB SSD • **5** — 512 GB SSD
CPU variant	This indicates the CPU variant of the device: • **4** — Intel E3940 (4 GB RAM) • **5** — Intel E3950 (8 GB RAM)

NOTE

Please make sure to write down the serial number of your Nerve Device. You are going to need it for the node registration process.

Front Panel Controls and Indicators

Below is an overview of the front panel of the MFN 100, describing physical interfaces, indicators and their labels.

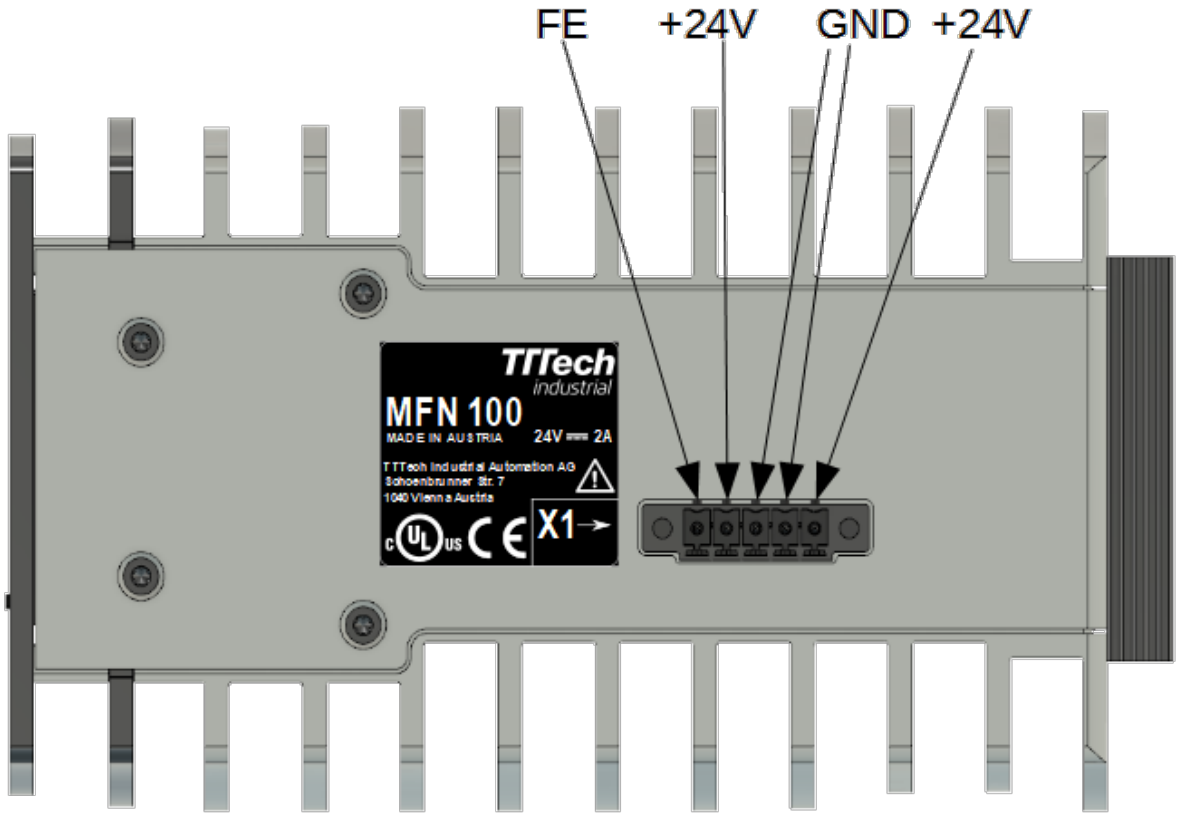


Label	Description
Connect Button	The connect button interrupts the connection on ports P2 to P5 of the MFN 100. This is the behavior in the standard configuration. The function is configurable on request. The button may be configured to change the network configuration.
Connection Indicator	The connection indicator is the first fin in the MFN 100 housing. It lights up blue when all required services are initiated and the connection to the Management System is configured.
Reset	Holding the button for 4-8 seconds initiates a power cycle. Use a tool with a rounded tip to press the button.
Power 1 **Power 2**	Indicators showing power active on the power supply.

Label	Description
Status	LED indicating system status • Green: All device functions are ready. • Not lit: Device functions are not ready or the device is booting.
Function	LED indicating CODESYS runtime status • Green: CODESYS runtime is operational. • Not lit: CODESYS runtime is not operational.
P1 Console	Ethernet port/console port. This port is typically used to connect a workstation to configure the MFN 100.
P2/P3/P4	Ethernet ports
P5	SFP port
I/O	Fieldbus interface
USB	Two USB 2.0 ports with 1.1 A maximum output current for both ports combined.
DP	Display Port supporting the DP++ standard.

Power Connectors Overview

The power connectors are located at the bottom of the MFN 100 next to the label. There are two separate 24 V inputs, two GND inputs and one Functional Earth (FE) input. The inputs are fused internally. The fuse cannot be replaced by the user. The power supply inputs are protected against reverse polarity.



Pin	Description
1	Power supply line 2
2	GND
3	GND
4	Power supply line 1
5	Functional Earth (FE)

NOTE

The GND and FE pins (pins 2, 3, and 5) are electrically connected to the housing.

Power Supply Details

Parameter	Value
Operating voltage	18 - 30 VDC
Start-up current	7 A max.
Consumption	1.4 A continuous 2.1 A peak
Dissipated power	33.6 W at 24 VDC

Installation and Removal on a DIN Rail

The MFN 100 is intended for mounting on a DIN rail inside a closed cabinet. Due to its weight it should be installed on a strong DIN rail. No tool is required to install or remove the MFN 100.

Follow these steps to install the MFN 100 on a DIN rail:

1. Engage the DIN rail mounting clip of the MFN 100 with the upper edge of the DIN rail.
2. Push the MFN 100 down into the DIN rail.
3. Place the MFN 100 in a vertical position so that the mounting clip engages the lower edge of the DIN rail.

If you want to remove the MFN 100 from a DIN rail, follow these steps:

1. Push the MFN 100 down.
2. Rotate the MFN 100 upwards so that the lower edge of the DIN rail disengages.
3. Lift the MFN 100 slightly to remove it.

Setting up the MFN 100

After mounting the MFN 100 on a DIN rail, it is ready to be set up as a Nerve Device. You will need two network cables and a +24 V DC power supply for the setup. This includes connecting the power supply to the mating connector which is delivered with the MFN 100.

1. Connect pin 1 of the mating connector to +24 V DC.
2. Connect pin 2 of the mating connector to GND.
3. Plug the mating connector into the bottom side of the MFN 100.
4. Connect port 2 of the MFN 100 to a DHCP-enabled network with access to the Management System or internet access if the Management System is hosted by TTTech Industrial.
5. Plug in the power supply.

The MFN 100 will start after a few minutes and light up blue when all necessary services are initiated.

NOTE

- If you are not sure how to allow external devices to connect to your network, please contact your IT administrator.
- If you want to connect the MFN 100 to a fieldbus, connect a network cable to the I/O port of the MFN 100 and to your fieldbus interface.
- You can also connect a second power supply to the MFN 100 as a backup. To do so, connect pin 3 of the mating connector to GND and connect pin 4 of the mating connector to +24 V DC.

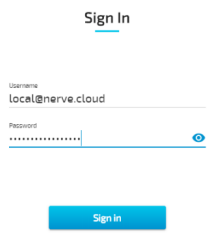
Local UI

As Nerve Devices have different hardware features, some configuration options differ as well. The main differences concern the local UI, networking and working with the CODESYS Development System. The local UI mostly changes according to the physical features of the Nerve Device.

Accessing the Local UI

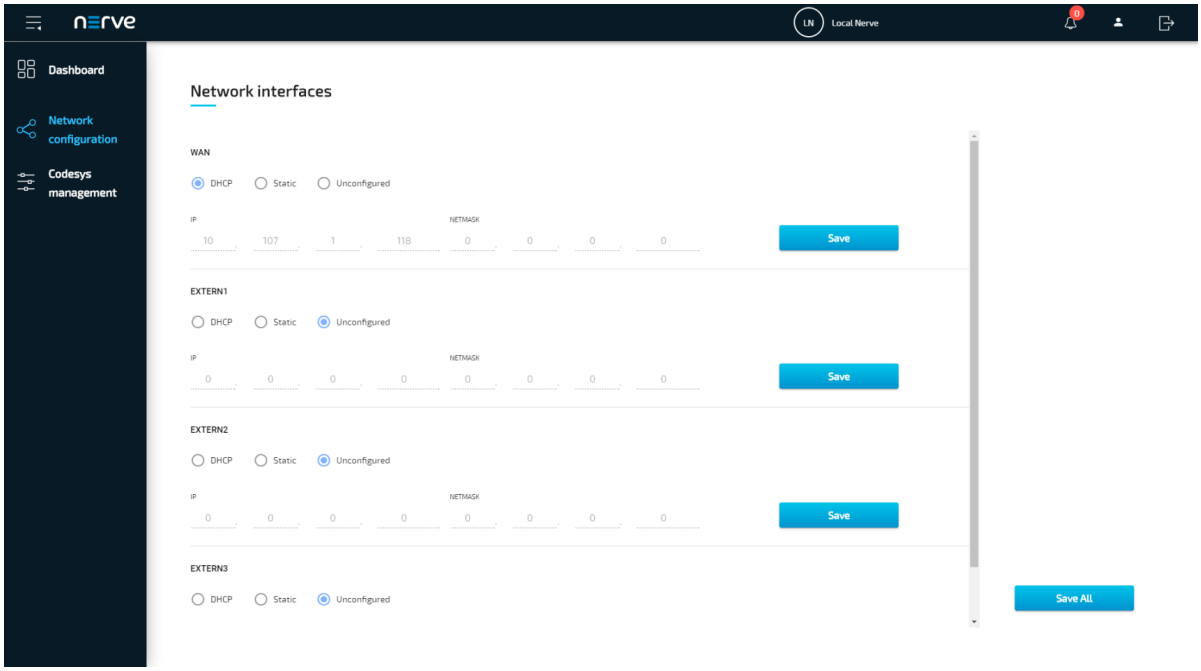
In order to access the local UI, you need to connect a workstation to the console port (P1) of the MFN 100 and configure the network adapter of your workstation. The IP address has to be in the range from 172.20.2.5 to 172.20.2.254 with a 255.255.255.0 subnet mask. You also need the credentials for the local UI that you can find in the customer profile.

1. Follow this link to connect to the local UI: http://172.20.2.1:3000/
2. Log in with the credentials from the customer profile.



Network Configuration

From the local UI, you can configure the Ethernet ports of the Nerve Devices. For the MFN 100, the interfaces in the local UI represent the physical ports 2, 3, 4 and 5. The console port (P1) and the I/O port of the MFN 100 are reserved and cannot be modified. The console port is used solely for configuration purposes. The I/O port is connected to the CODESYS runtime and used for fieldbus communication. Select **Network configuration** in the navigation on the left to reach this menu.



Available Network Interfaces

Interface Name	Physical Port on the MFN 100
WAN	Ethernet port 2 (P2)
EXTERN1	Ethernet port 3 (P3)
EXTERN2	Ethernet port 4 (P4)
EXTERN3	Ethernet port 5 (P5)

For more information on the network interfaces and their application, please see the networking chapter in the user guide.

CODESYS Related Information

For working with the CODESYS Development System, a device description per Nerve Device is required. The device description of the MFN 100 has been sent as part of the delivery. Please contact TTTech Industrial customer support or your sales representative if that is not the case.

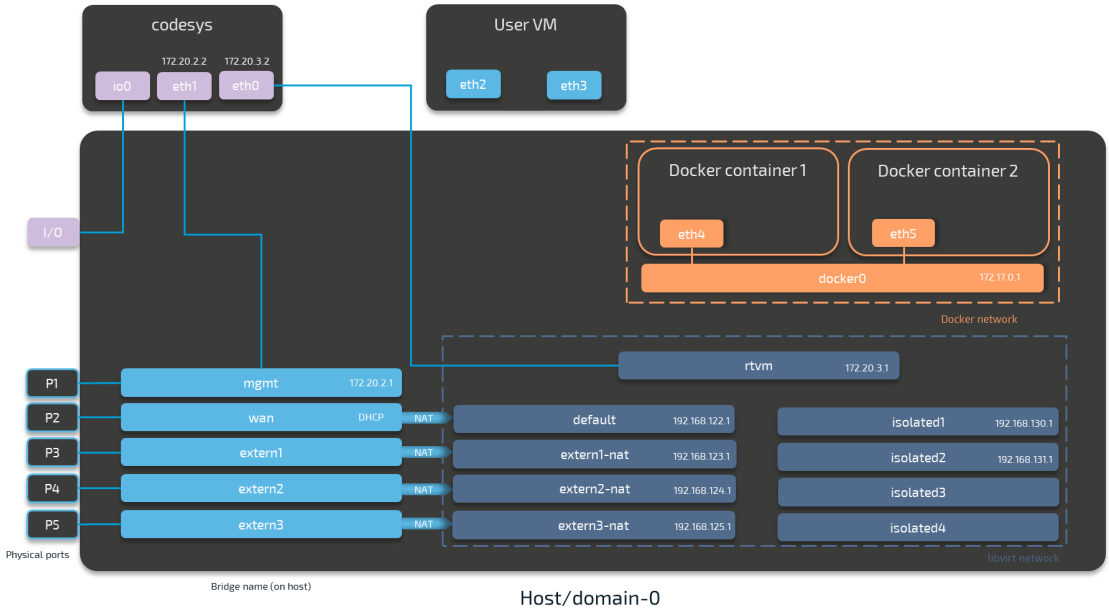
The MFN 100 has an Ethernet port that is reserved for machine data acquisition. Connect a network cable to the I/O port of the MFN 100 and to your fieldbus interface to acquire machine data. The CODESYS runtime can be reached at 172.20.2.2.

Networking

Below is a graphic that details the available interfaces of the MFN 100 for use with Nerve Blue. Pictured is how the physical interfaces translate to the Host and the CODESYS runtime.

The image below shows an example node consisting of the **host/domain-0** and the real-time VM running the CODESYS runtime (labeled **codesys**). It also has one Virtual Machine workload and two Docker workloads deployed. The virtual machine is located outside of the host and the Docker containers are located in the Docker network inside of the host. However, the workloads are not yet connected.

Notable IP Adresses	
Host access	172.20.2.1
CODESYS runtime access	172.20.2.2

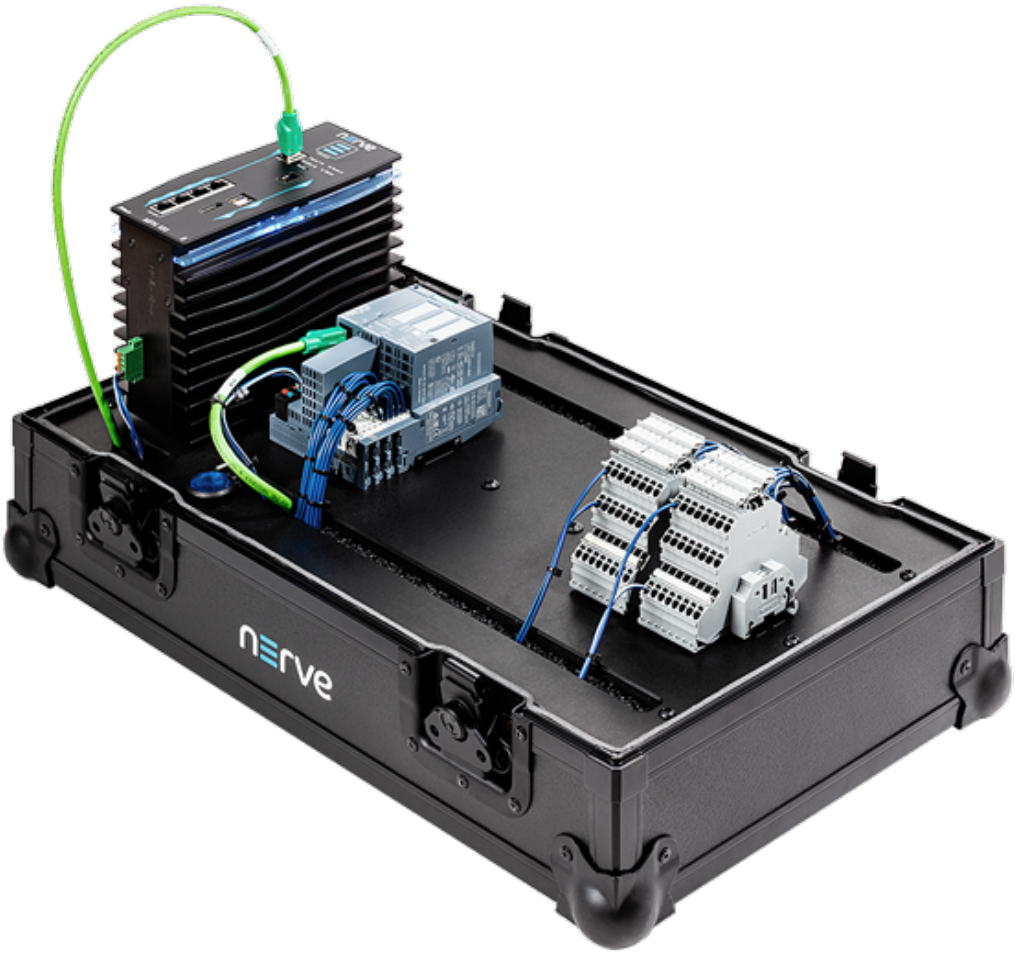


Please refer to the networking chapter in the user guide for more information on networking in the Nerve Blue system.

Release Notes

Nerve Blue Kit

Nerve Blue Kit



The Nerve Blue Kit is a ready-to-use hardware and software package, designed to support the demonstration and evaluation of Nerve Blue features. The kit provides everything needed to start collecting, storing and analyzing data, and can be expanded to include your own machine infrastructure. Nerve Blue's virtualized software environment enables users to run multiple container- or virtual machine-based applications and to manage them remotely. In addition, the kit can be used to connect to real-time data sources and update real-time control applications remotely via the Management System.

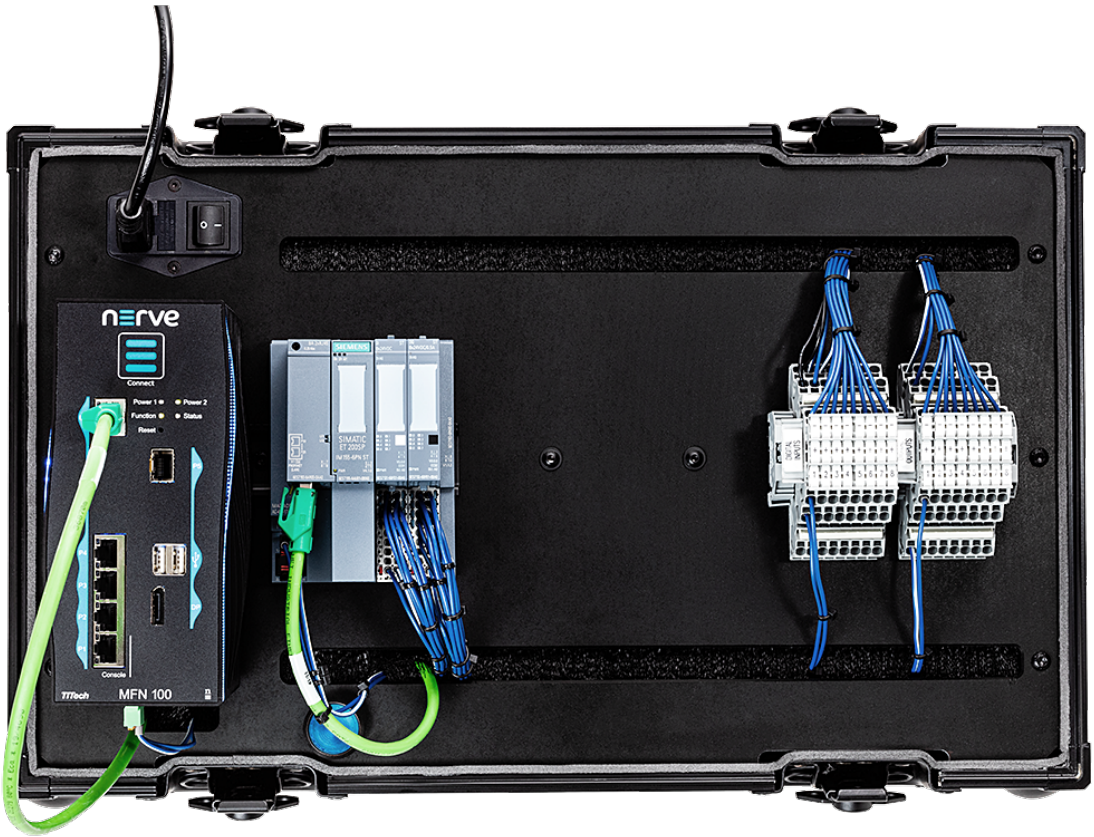
Key Features

The Nerve Blue Kit is fully integrated in a robust, portable case suitable for lab use and demonstrations. Start with default control applications, then develop and deploy your own and even extend the kit's functionality by adding your own sensors and actuators.

Plug and play	Extendable
• Two pre-installed CODESYS control applications for demo purposes • Workloads (Node-RED container / CODESYS programs) available for deployment from the Management System to the Nerve Device	• Extra digital I/Os available for adding your own sensors to extend functionality • Newly created workloads (virtual machines / containers / CODESYS programs) can be deployed from the Management System to the Nerve Device

Nerve Blue Kit Contents

The contents of the kit are delivered in two separate boxes, a small box and a large box. The small box contains the MFN 100 including the Hardware Installation Guide and a mating connector. The large box has the remaining components of the kit: the mounting plate, the SIMATIC I/O module and the digital I/O terminal block.



Please note that you will also need a power cord and two network cables to finish the setup.

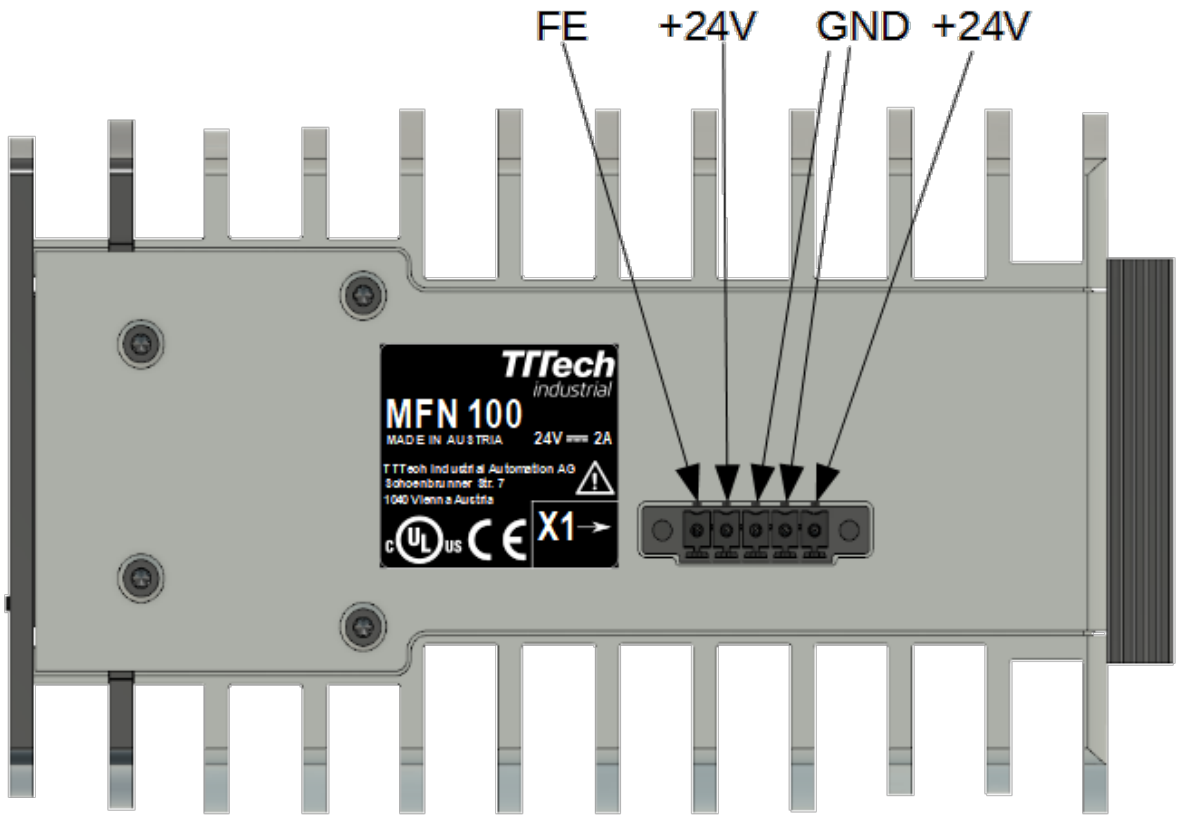
Content Overview

Hardware	Software

• MFN 100-C64xx
• SIMATIC ET200SP I/O system including:
  1x SIMATIC ET 200SP bundle PROFINET IM, IM 155-6PN ST
  1x SIMATIC ET 200SP 8x24V DC digital input
  1x SIMATIC ET 200SP 8x24V DC digital output
• Digital I/O terminal block
• Illuminated push button
• 24V DC Power supply
• 1x Mating connector
• 1x Network cable
	• MFN 100-C64xx device description file
	• SIMATIC ET200SP I/O device description file
	• Two default CODESYS projects

Power Connectors Overview

The power connectors are located at the bottom of the MFN 100 next to the label. There are two separate 24 V inputs, two GND inputs and one Functional Earth (FE) input. The inputs are fused internally. The fuse cannot be replaced by the user. The power supply inputs are protected against reverse polarity.



Pin	Description
1	Power supply line 2
2	GND
3	GND
4	Power supply line 1
5	Functional Earth (FE)

NOTE

The GND and FE pins (pins 2, 3, and 5) are electrically connected to the housing.

Setup and Default Operation

This section will walk you through the hardware setup, explain the default operation of the kit and give an introduction to the Management System.

Setting up the Hardware

The MFN 100 is delivered in a separate box and has to be mounted on the kit. A mating connector is already connected to the power supply. Also, the SIMATIC ET 200SP I/O module is already connected to the I/O port of the MFN 100 with a network cable. In order to completely set up the kit, you will need a power cord and two network cables.

1. Mount the MFN 100 on the DIN rail on the left side of the kit.

   NOTE

   If you need help with mounting the MFN 100 on the DIN rail, please refer to the Hardware Installation guide enclosed in the box of the MFN 100.

2. Plug the mating connector that is connected to the power supply into the bottom side of the MFN 100.

3. Connect port 2 of the MFN to a DHCP-enabled network with access to the Management System or internet access if the Management System is hosted by TTTech Industrial.

   NOTE

   If you are not sure how to allow external devices to connect to your network, please contact your IT administrator.

4. Connect the power cord to the power supply and to a power outlet. Make sure the power supply is turned off.

5. Push the button to switch on the power supply.

The MFN 100 will start after a few minutes and the blue light will turn on. All necessary services are initiated and after that, data is sent to the Management System.

Default Operation

The Nerve Blue Kit comes with two default applications which can replicate scenarios found in industrial automation. The applications allow for data to be generated at the I/O, be acquired by the MFN 100 via Ethernet fieldbus, translated to OPC UA and be sent to the cloud for visualization. The data can be generated as a continuous flow throughout the cycle, as a regular data flow, or as irregular event based data flow. These different data flows mimic various types of industrial process and show data flow from I/O to cloud.

The applications use the push button that is connected to the SIMATIC ET200 SP I/O module via the terminal block. Data is sent between the SIMATIC I/O module and the MFN 100 I/O port via PROFINET and the illumination of the button is controlled via the CODESYS soft PLC running on the MFN 100.

The two default CODESYS applications: **app1** and **app2** are used to alter flow of data from the I/O modules. Both apps control the illumination of the button and record two values: iCountNumber and iCountButton.

	app1	app2
iCountNumber (continuous data flow from the application to the Management System)	• Continuous count upwards from 0 to 1000. • Resets to 0 automatically when it has reached 1000.	• Continuous count downwards from 1000 to 0. • Resets to 1000 automatically when it has reached 0.
iCountButton (irregular and regular data flow from the application to the Management System)	• Counts the number of times that the button has been pressed (irregular data flow). • The push button lights up after 10 button presses. • The value resets to 0 when the button is held for two seconds. The light of the button goes out.	• Counts the duration that the button is held (regular data flow). • The counter continuously increases from 0. The push button lights up when the value reaches 25. • The value continuously decreases to 0 when the button is released.

The Management System allows users to deploy either of the default applications from the repository to the MFN 100. This demonstrates the ability to update CODESYS applications remotely and alter applications running on machines wherever they are in the world.

Beyond the two default applications, users can also use the kit to develop their own CODESYS applications, which can be uploaded to the repository in the Management System and then deployed to the MFN 100. These applications can then be used in conjunction with corresponding actuators and/or sensors.

NOTE

The default application **app1** is loaded automatically during the initial startup of the MFN 100.

Connecting to the Management System

The Nerve Management System is a web-based service that permits management of Nerve Blue nodes that are registered.

NOTE

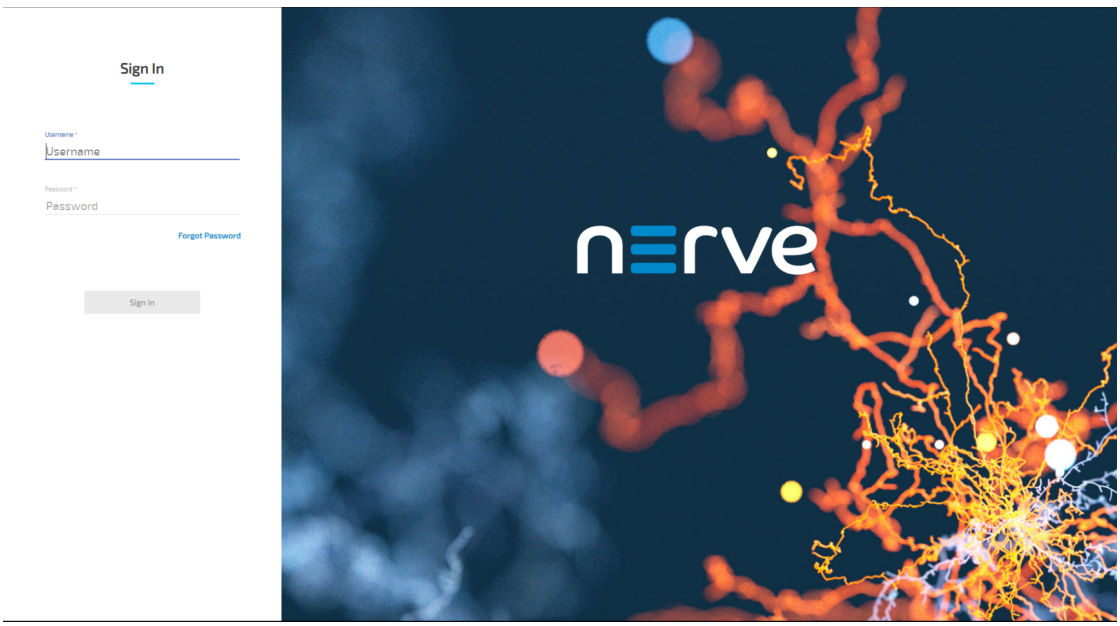
Google Chrome or Firefox Version 63 or later are recommended for the usage of the Management System.

Before connecting, make sure that the MFN 100 of the starter kit is connected to the network through port 2 and that an IP address has been assigned by the DHCP server. If you need help with assigning an IP address, please contact your IT administrator.
The login credentials for the Management System are in the customer profile. The customer profile has been sent in form of a PDF as part of the delivery.

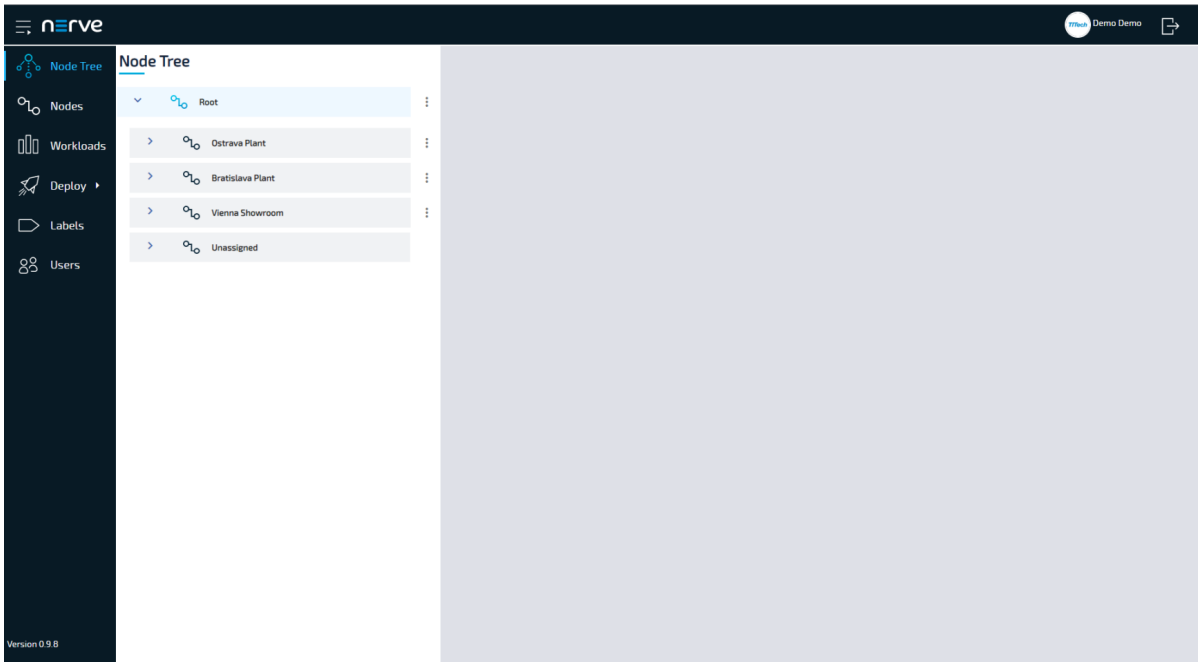
NOTE

If you have not received a customer profile, please contact your sales representative or TTTech Industrial customer support at support@tttech-industrial.com.

1. Go to the URL of the Management System in the customer profile.
2. Log in with the credentials provided in the customer profile.



You will reach the dashboard of the Management System, which shows the Node Tree by default.



One element in the Node Tree is already created at first login, containing the MFN 100 of the Nerve Blue Kit. All newly registered nodes will be located under **Root > Unassigned** by default.

From here you can manage nodes, provision workloads and deploy workloads among other options. See the user guide for more information on the Management System.

NOTE

Port 443 (HTTPS) and port 8883 (MQTTS) of the corporate firewall have to be open for communication between nodes and the Management System.

Moving a Node from One Tree Element to Another

Moving nodes in the Node Tree is possible by drag and drop. Please make sure that you have created a new tree element before attempting to move a node.

1. Select **Node Tree** in the left-hand menu.
2. Expand the tree element of the node you want to move. The default element is **Root > Unassigned**.
3. Choose the node you want to move.
4. Drag and drop the node to your newly created element. Elements expand automatically once you hover over them.

5. Select **APPLY CHANGES (n)** in the upper-right corner of the Node Tree.

NOTE

(n) is a placeholder for the number of changes made to the Node Tree. If you have performed three changes, you will see (3) in the button above the Node Tree.

The node has now been moved to the new element. Note that a node cannot be moved back to **Unassigned** once it has been moved to another element.

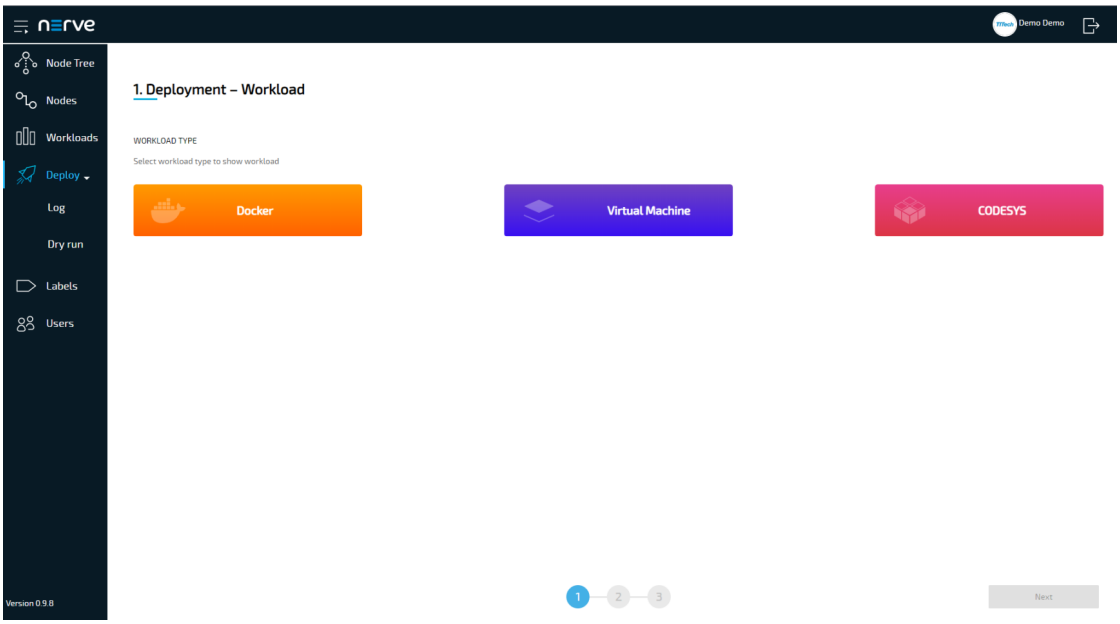
Downloading & Uploading CODESYS Applications from the Management System

With the kit you can deploy CODESYS applications from the Management System to the MFN 100. The two default CODESYS applications are already available in the Management System. You can also upload your own applications to the repository in the Management System and make them ready for deployment.

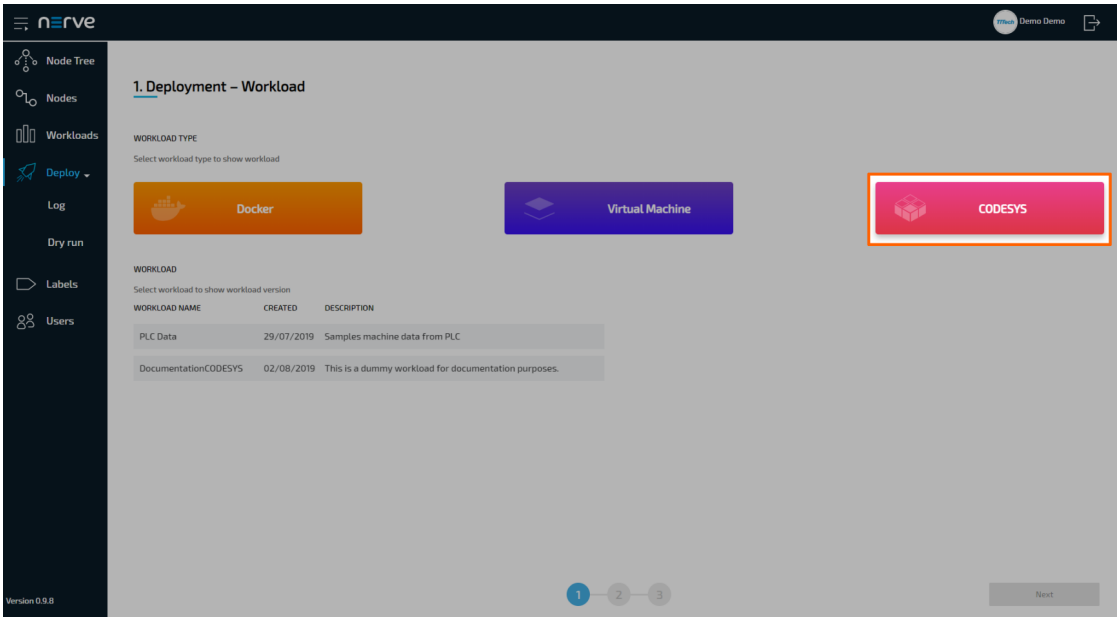
Deploying a CODESYS Workload

One CODESYS workload is available with first login that can be used with the kit immediately. The workload is named **Nerve Blue Kit** and it has two versions: **Nerve Blue Starter Kit App1** and **Nerve Blue Starter Kit App2**. For more information on the deployment process for all workload types refer to the user guide.

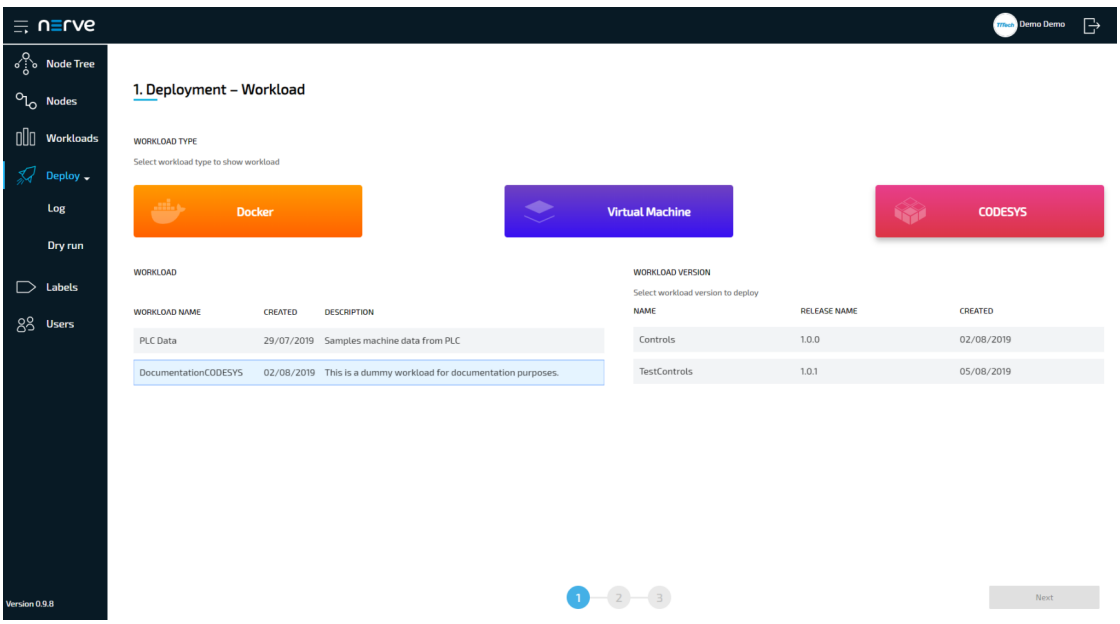
1. Select **Deploy** in the left-hand menu.



2. Select the CODESYS workload icon on the right. A list of CODESYS workloads will appear below.

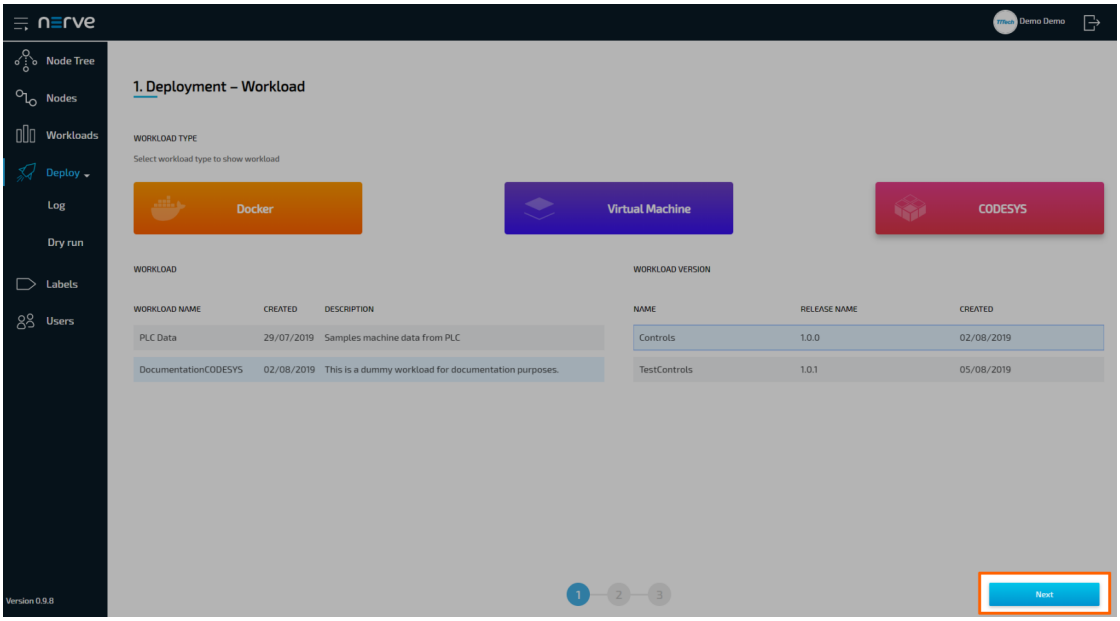


3. Select a workload from the list. A list of versions of this workload will appear to the right.



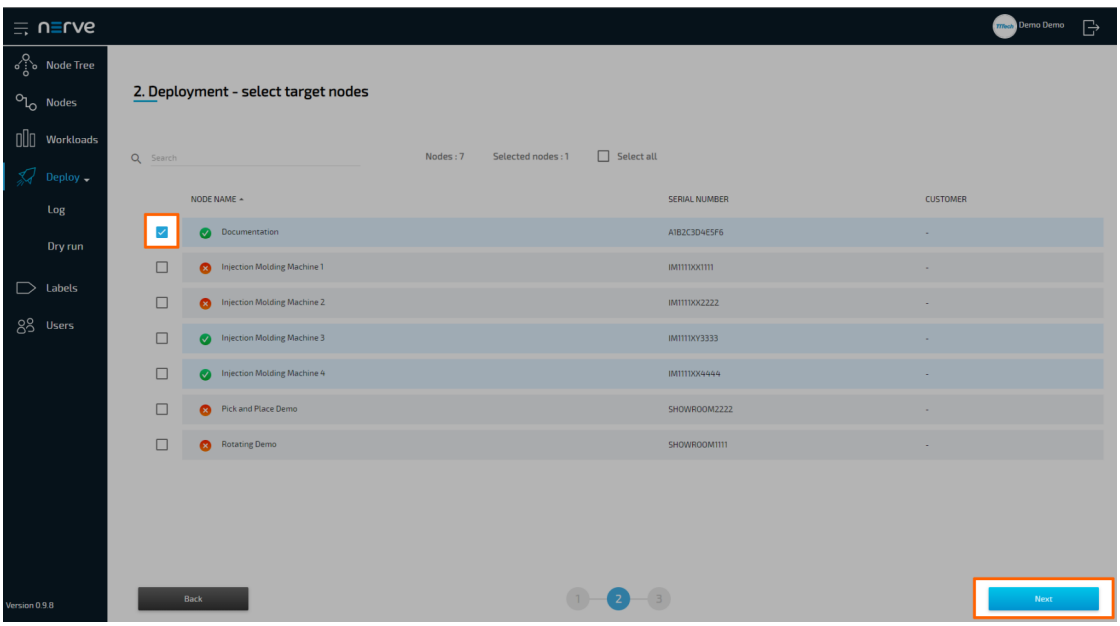
4. Select the version of the workload you would like to deploy.

5. Click **Next** in the bottom-right corner.



6. In the next window, select one or more nodes from the list for deployment by ticking the checkboxes on the left.

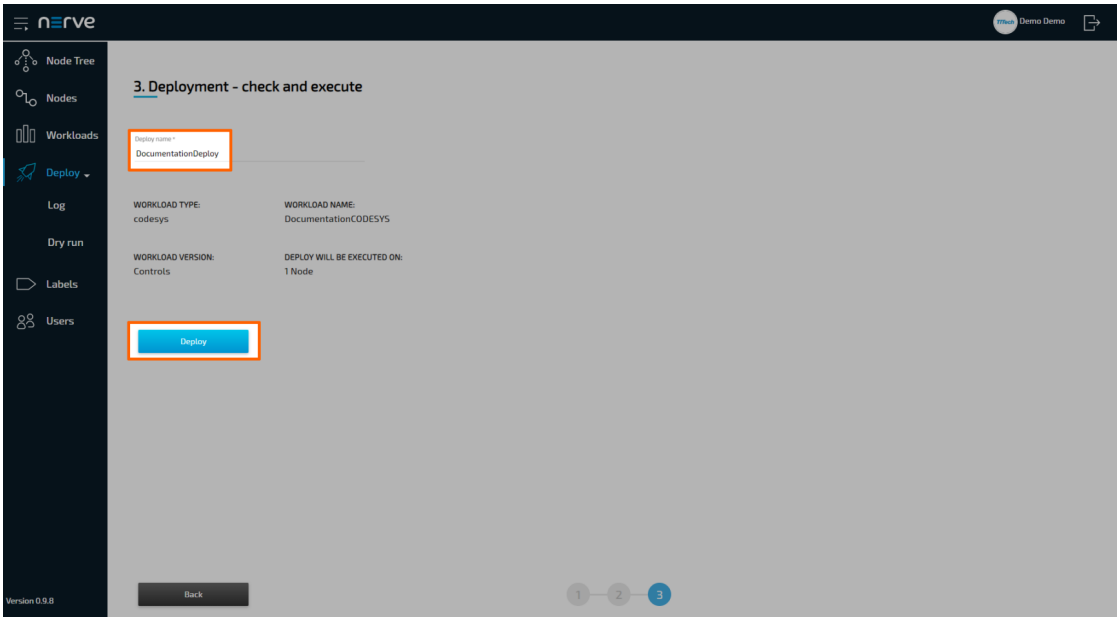
7. Select **Next** in the lower-right corner.



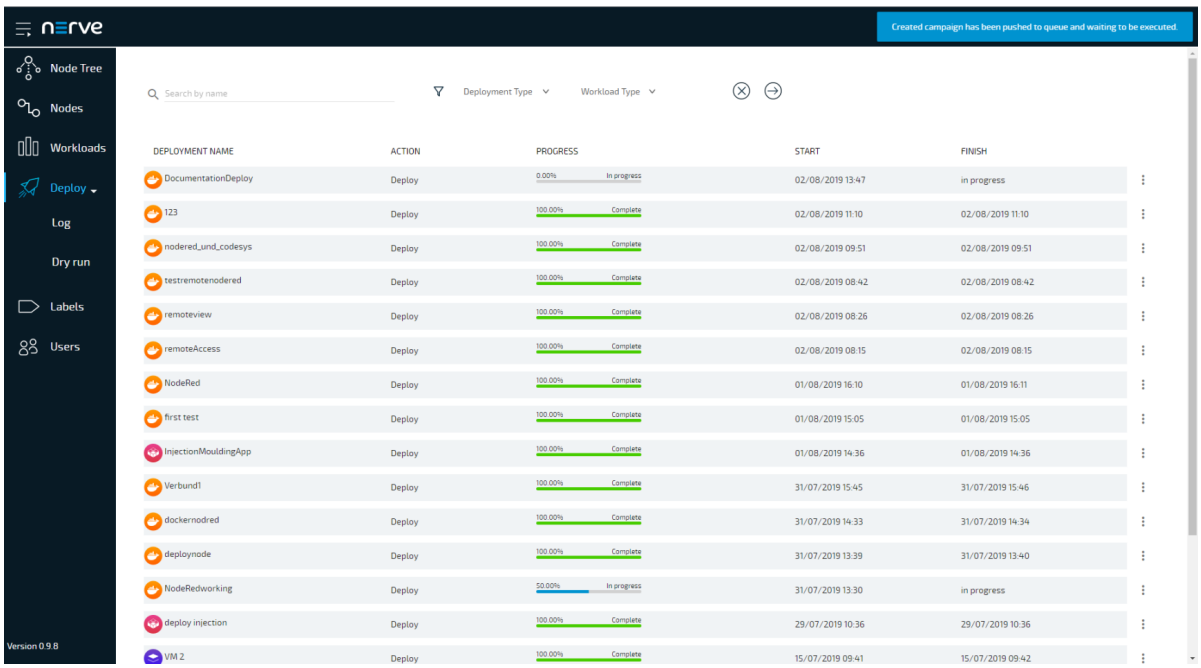
8. In the next window, enter a **Deploy name** above the **Summary** of the workload. Choose a name that makes this deployment easy to identify.

> **NOTE**
>
> You can only use alphanumeric characters (a-z, A-Z, 0-9) and underscore (_). Any other special characters are not allowed. If you use a character that is not allowed, the deploy button will be grayed out.

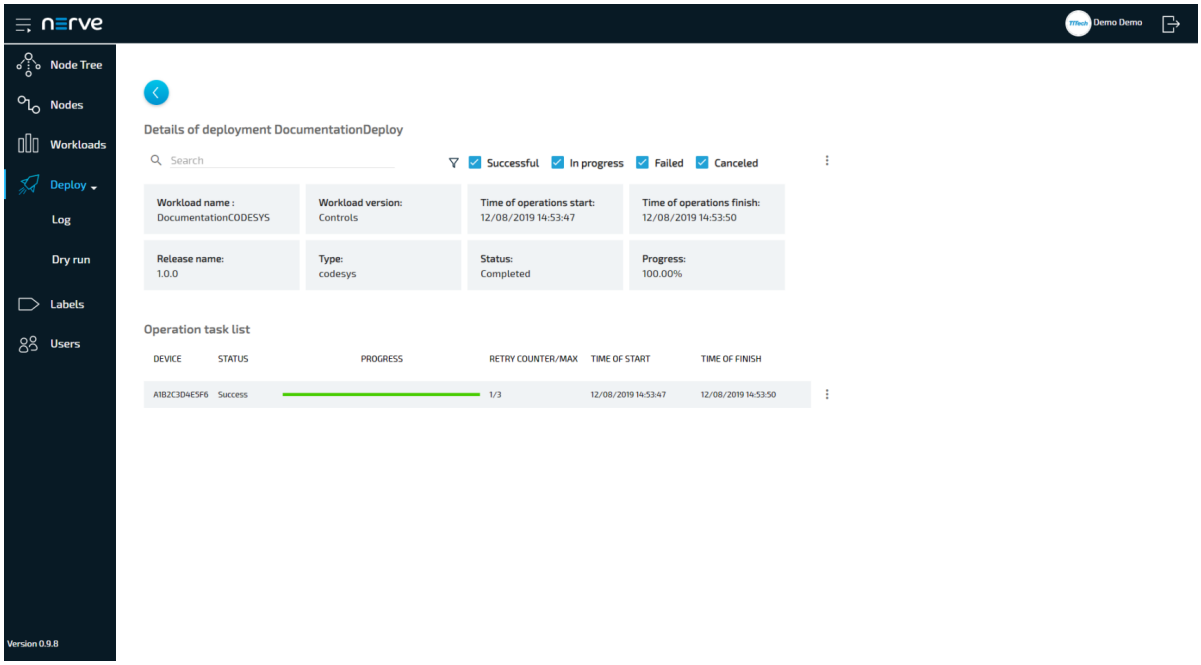
9. Select **Deploy** to execute the deployment.



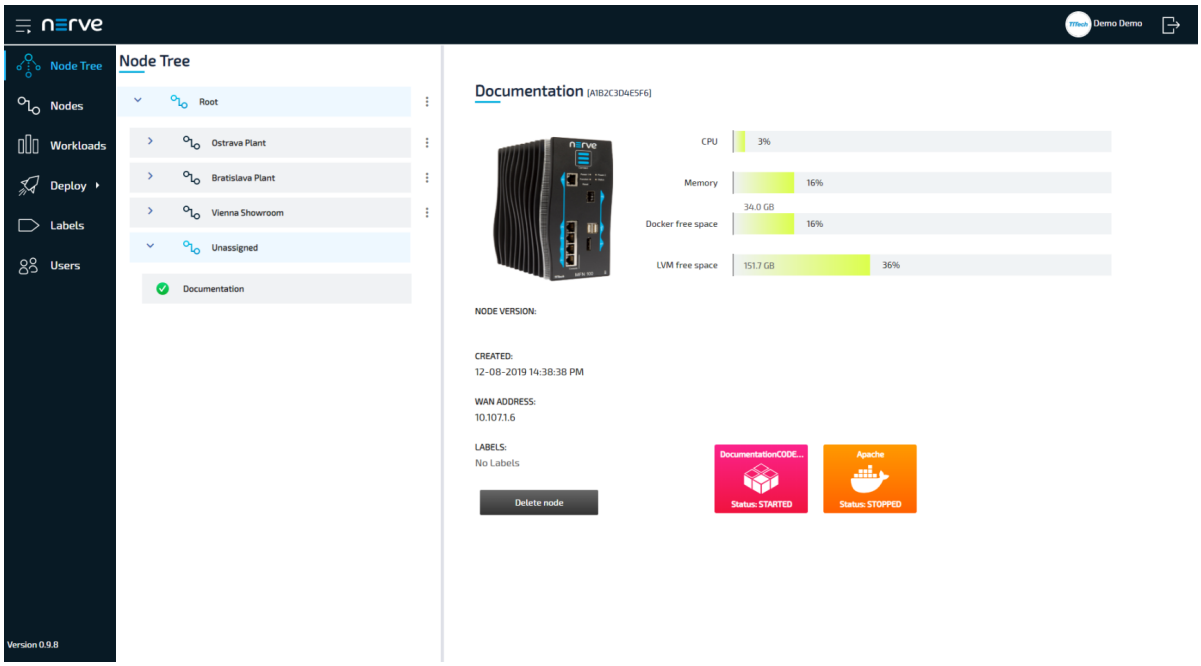
You will be taken to the log next. Your current deployment is at the top of the list. The**Deploy name** you have chosen before is the name that identifies the deployment in the log.



You can see the progress of the current deployment and click the workload to see a more detailed view.



Confirm the deployment by viewing the workload in the node details view in the Node Tree. Select **Node Tree** in the menu on the left and select the node you have deployed a workload to.



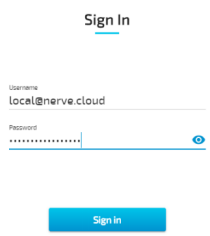
Clicking a workload tile leads to the workload control screen. This is where workloads can be controlled. However, CODESYS workloads can only be controlled through the local UI.
All workloads are started as soon as they are deployed.

Connecting to the Local UI

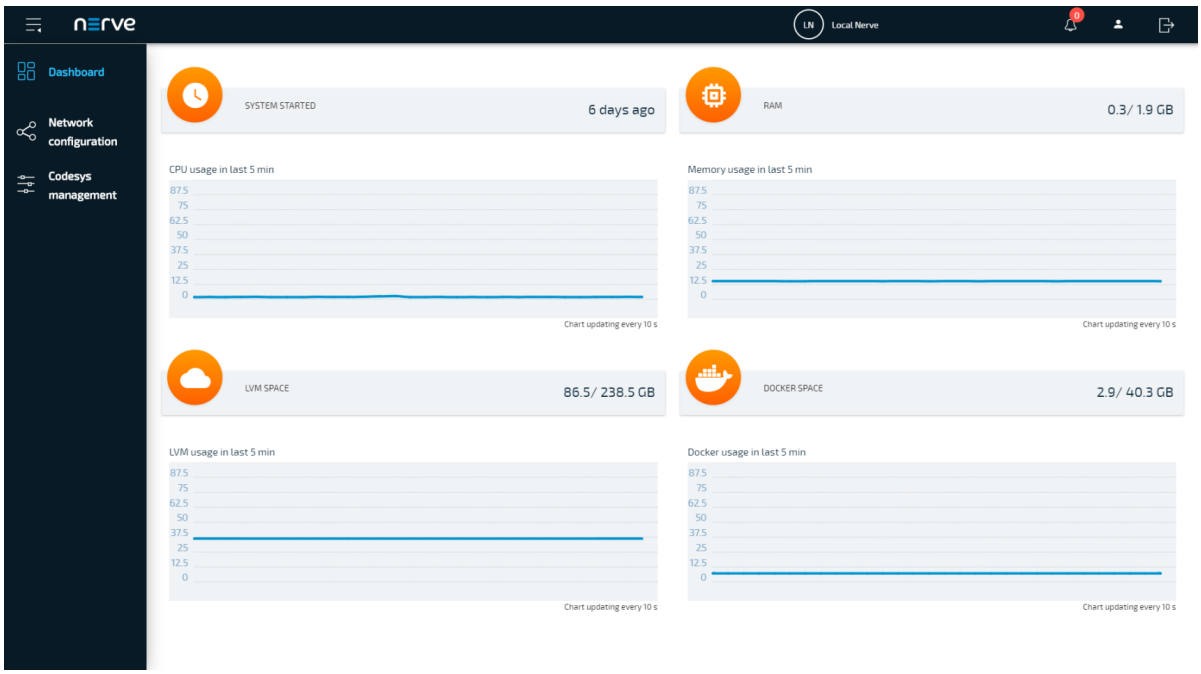
In order to access the local UI, you need to connect a workstation to the console port (P1) of the MFN 100 and configure the network adapter of your workstation. The IP address has to be in the

range from 172.20.2.5 to 172.20.2.254 with a 255.255.255.0 subnet mask. You also need the credentials for the local UI that you can find in the customer profile.

1. Follow this link to connect to the local UI: http://172.20.2.1:3000/
2. Log in with the credentials from the customer profile.

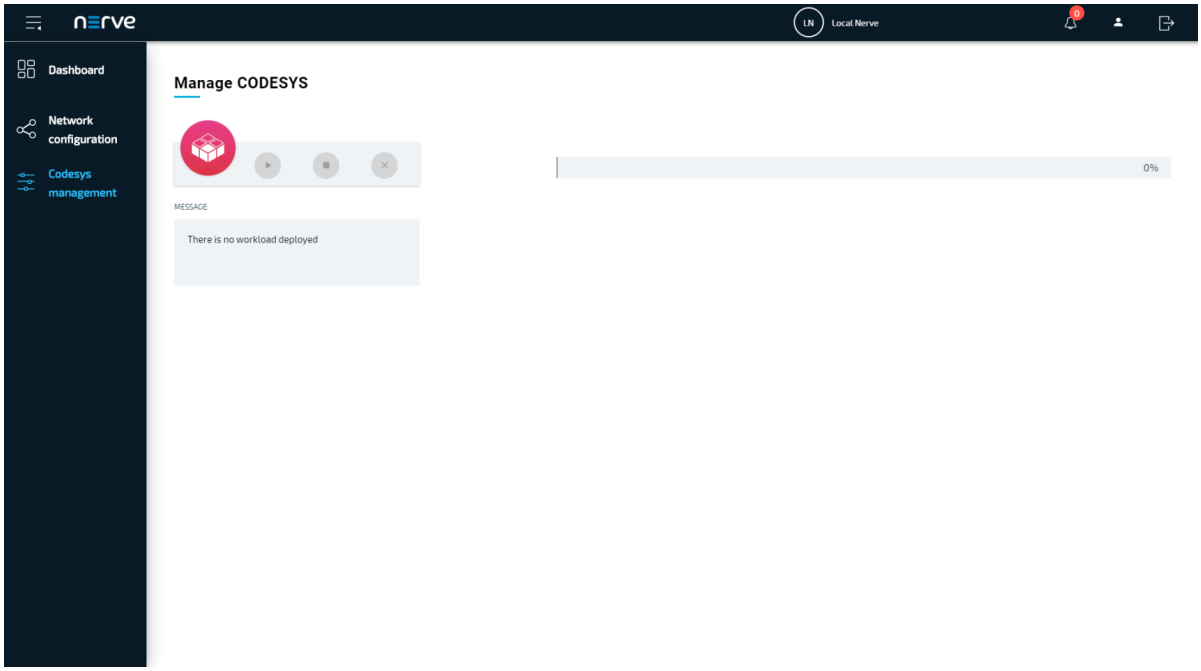


You will reach the main page of the local UI.



Control of CODESYS Applications

CODESYS workloads can only be controlled in the local UI, as operation of a CODESYS workload may have an impact on your machine operation and therefore should not be controlled remotely. Select **CODESYS management** in the menu on the left-hand side to reach the interface for controlling a CODESYS application running on the Nerve Device:



Function Name	Description
Start	This starts the CODESYS application.
Stop	This stops the CODESYS application and it is reset to its initial values.
Remove	This removes the CODESYS application from the Nerve Device. If you want to deploy the CODESYS application again, you have to do so through the Management System in the cloud.
Message	CODESYS workloads have the following set of messages: • "Preparing files for installation" • "Starting CODESYS application" • "CODESYS application started" • "Stopping CODESYS application" • "CODESYS application stoppped" • "Removing CODESYS application file" • "An unexpected error has occurred. <errormessage>" Here, <errormessage> is a message that is sent by the CODESYS Development System.

NOTE

It takes a moment before CODESYS applications are actually started, stopped or removed.

Uploading New CODESYS Applications

In order to work with new CODESYS applications on the MFN 100, new CODESYS workloads need to be provisioned in the Management System. Here, provisioning is the creation of a workload and its storage in the workload repository of the Management System so that it can be deployed to nodes.

This requires configuration of the workload and the upload of the required files to the Management System. After that, the workload can be deployed to nodes.

Before the workload can be provisioned, however, a CODESYS application has to be loaded into the CODESYS runtime first. Refer to the introduction to working with CODESYS and the MFN 100 first before continuing.

> **NOTE**
>
> Note that **app1** is already loaded into the CODESYS runtime by default. If you wish to try the provisioning process, continue in the current chapter.

Once a CODESYS application has been loaded into the MFN 100, the following steps have to be taken before provisioning a CODESYS workload:

1. Creating the ZIP file of the CODESYS application
2. Transferring the ZIP file to a local workstation

In this version, Nerve Blue does not provide a GUI based method for creating a ZIP file of the CODESYS application. Therefore, this chapter focuses on the manual process. Two tools are required for the instructions below, assuming Windows is used on the workstation:

- an SSH client like PuTTY
- a file transfer client like WinSCP

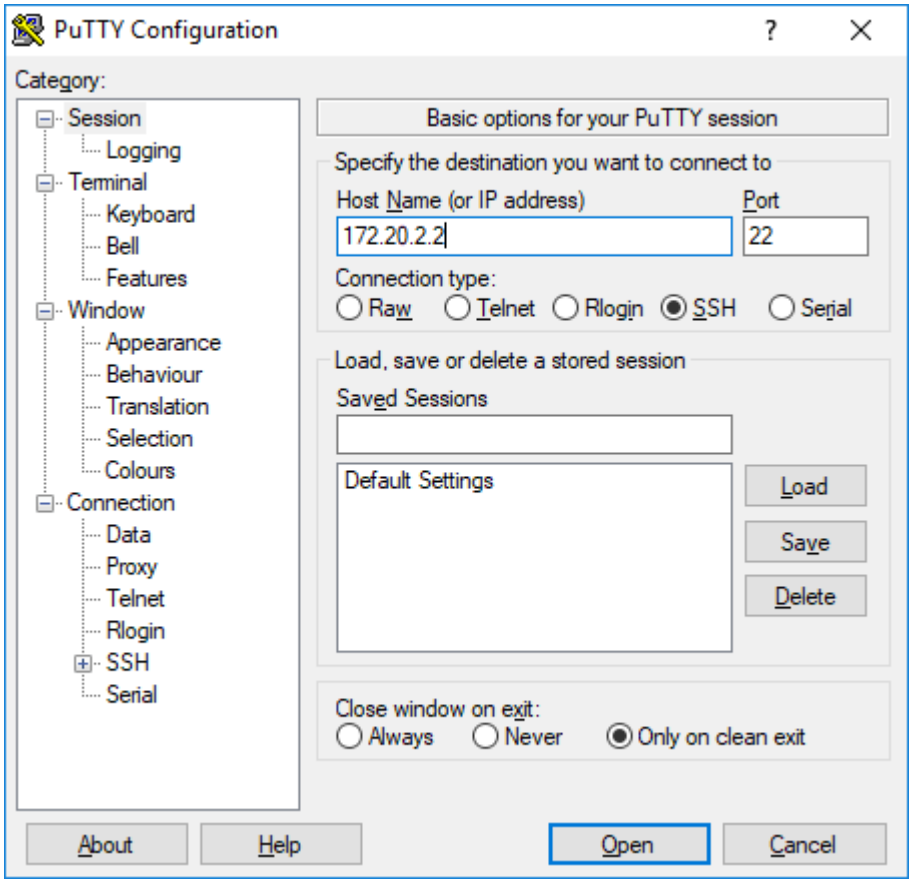
Also the workstation needs to be connected to the console port (P1) of the MFN 100 and the network adapter of the workstation needs to be configured. The IP address has to be in the range from 172.20.2.5 to 172.20.2.254.

Creating the ZIP File on the Nerve Device

First, the CODESYS project needs to be zipped on the Nerve Device before it can be copied from the CODESYS runtime.

1. Open an SSH client like PuTTY.

2. Enter 172.20.2.2 under **Host Name (or IP address)** to log in to the CODESYS runtime of the MFN 100.



3. Click **Open**.
4. In the new window, enter the admin password for the CODESYS runtime.
5. Enter the following commands:

   sudo su

   systemctl stop nerve-codesys

   cd/var/lib/codesys

   zip -r /tmp/<userproject>.zip CmpApp.cfg PlcLogic/

> **NOTE**
>
> You can freely name the ZIP file here by replacing <userproject> with a name of your choice.

The ZIP file has now been created and is available on the Nerve Device.

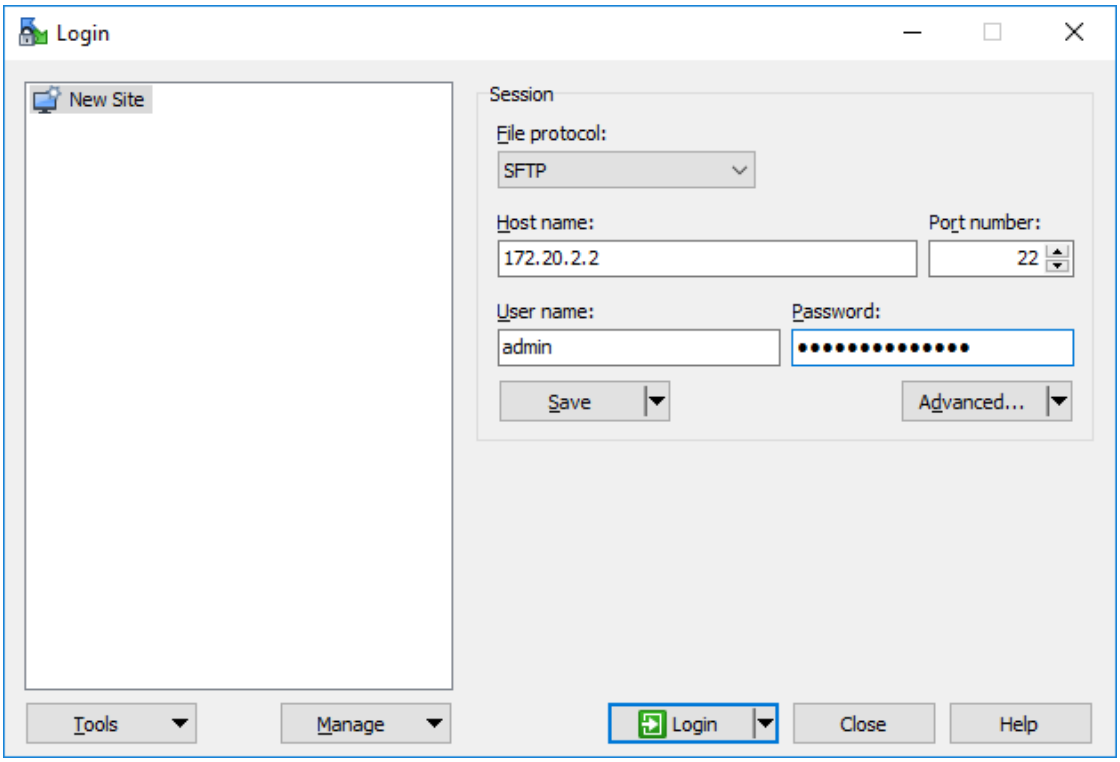
Transferring the ZIP File to a Local Workstation

The zipped CODESYS project file needs to be transferred from the MFN 100 to a workstation with a file transfer client in order to be uploaded to the Management System.

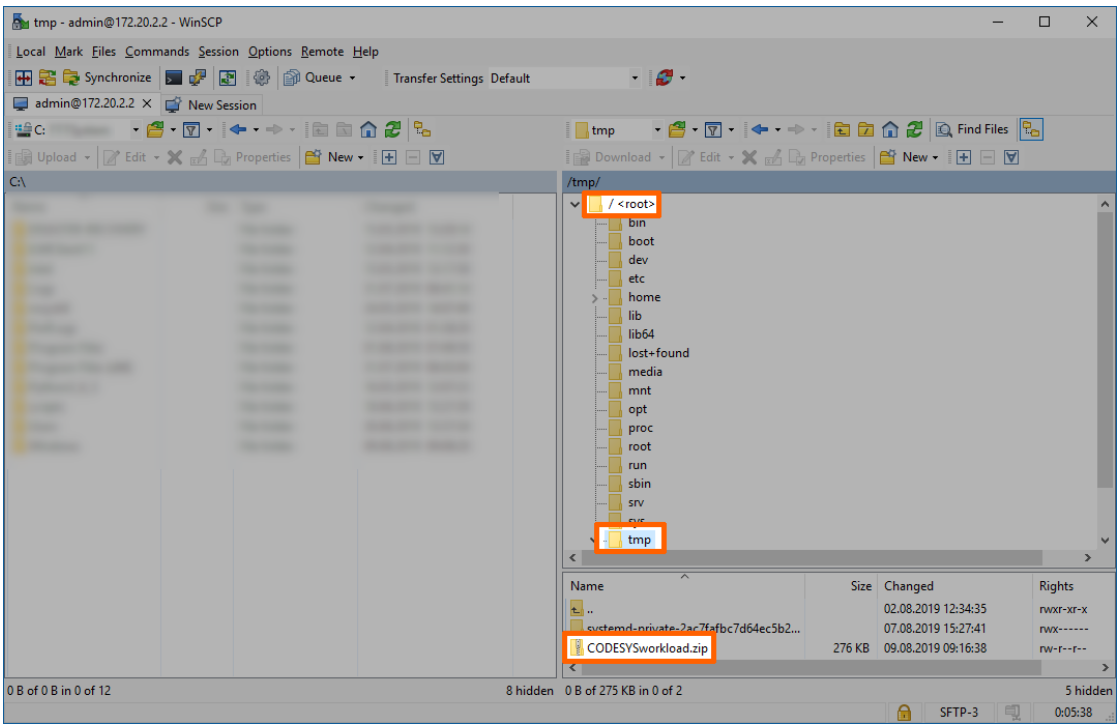
1. Open a file transfer client like WinSCP.

2. Enter 172.20.2.2 under **Host Name**.

3. Enter the credentials for the CODESYS runtime below under **User name** and **Password**.



4. Navigate to the **/tmp/** directory on the right side of the WinSCP window. You will find it in the **root** directory.



5. Drag and drop the ZIP file to your workstation.

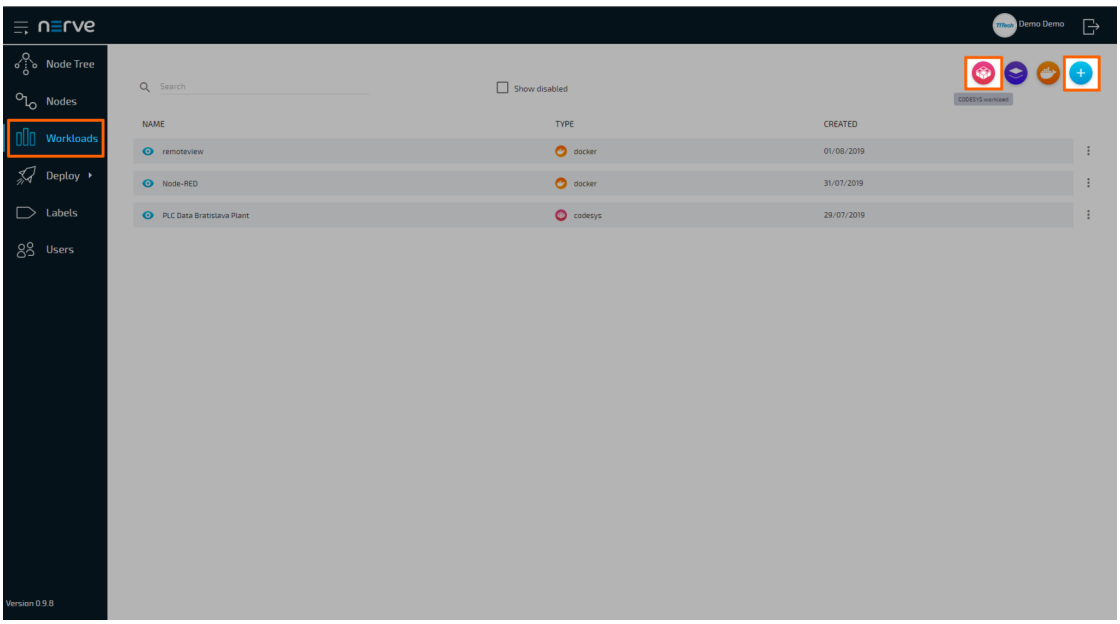
Since the CODESYS project ZIP file is on your local workstation, you can now provision a CODESYS workload in the Management System.

Provisioning a CODESYS Workload

The following instructions cover the basic requirements for provisioning a CODESYS workload. Optional settings will be left out. Extended options are addressed in the user guide.

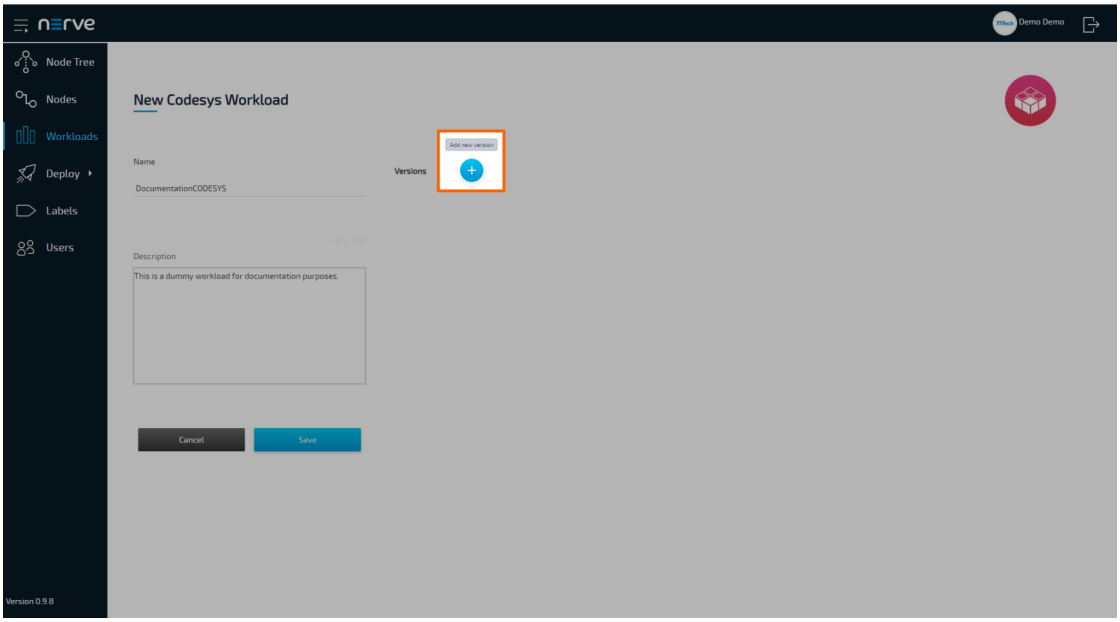
1. Log in to the Management System.
2. Select **Workloads** in the left-hand menu.
3. Select **+** in the upper-right corner.

4. Select the CODESYS symbol (**CODESYS workload**) on the left of the three symbols that expanded.



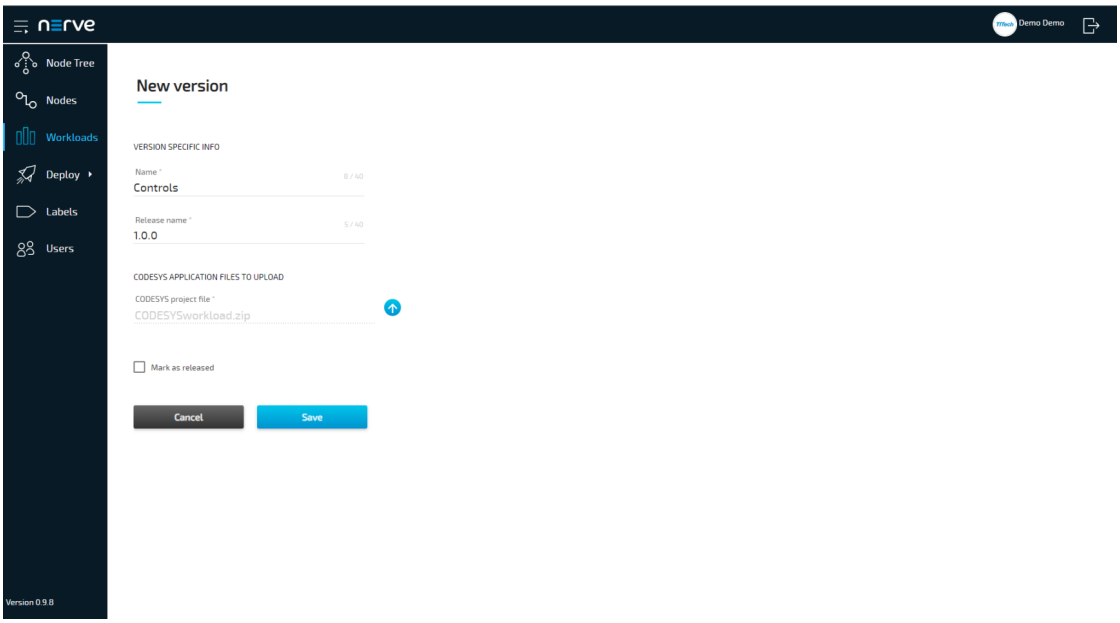
5. Enter a name for the workload in the new window.

6. Select **+** next to **Versions** to add a new version of the workload.



7. Enter the following information in the new window:

Item	Description
Name	Enter a **Name** for the version of this workload.
Release name	Enter a **Release name** for the version of this workload.
CODESYS project file	Click the **upward arrow** symbol to open the file browser and add the CODESYS application ZIP file. This is the ZIP file that you have created before.



8. Click **Save**.

The workload has now been provisioned and is ready to be deployed in the **Deploy** menu.

Connecting New Sensors and Actuators

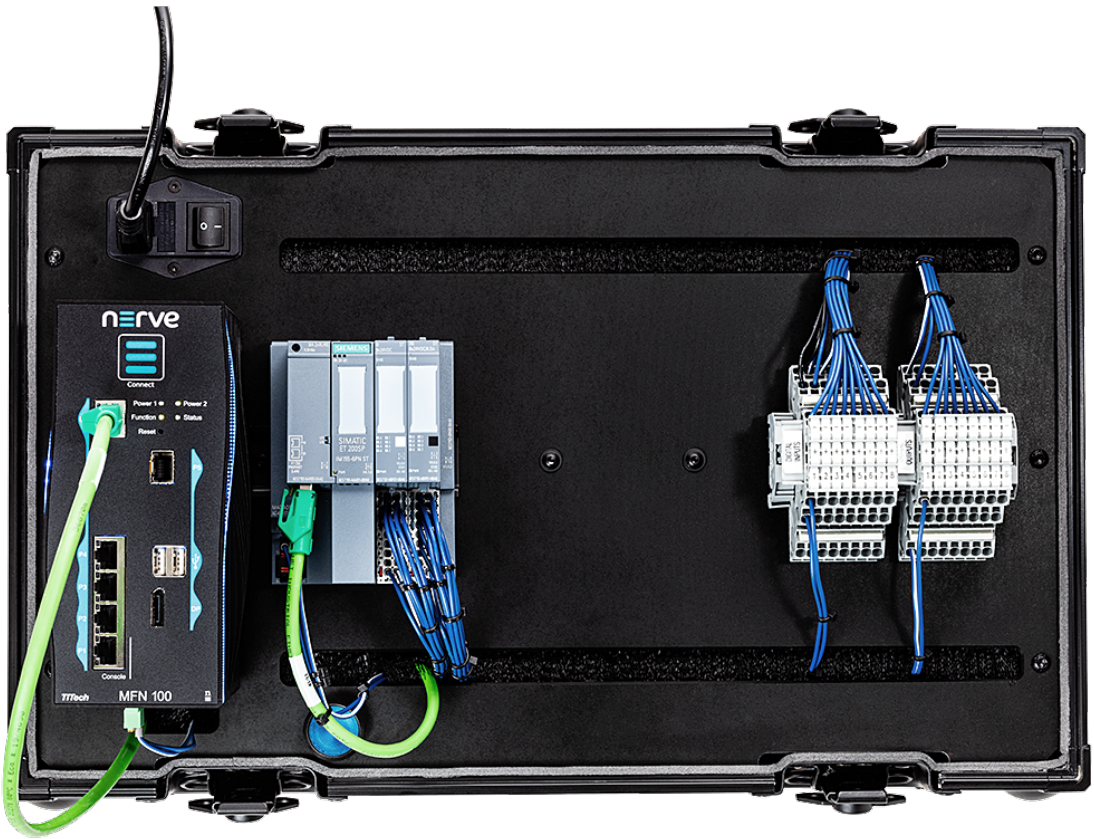
With the kit you can also add your own sensors and actuators to execute control applications and visualize the corresponding data. With the delivered set-up it is possible to add up to 7 additional inputs and outputs. You can also add your own I/O blocks to increase the number or type of inputs and outputs.

Wiring a New Sensor or Actuator

NOTE

- Before wiring any new components please review the Nerve Blue Kit Circuit Diagram that can be found in the service area or contact a sales representative.
- Disconnect the power supply from the power outlet before wiring new I/O devices to prevent injury to persons or damage to equipment.
- Only staff with knowledge about electrical circuits should perform the tasks described in this section.

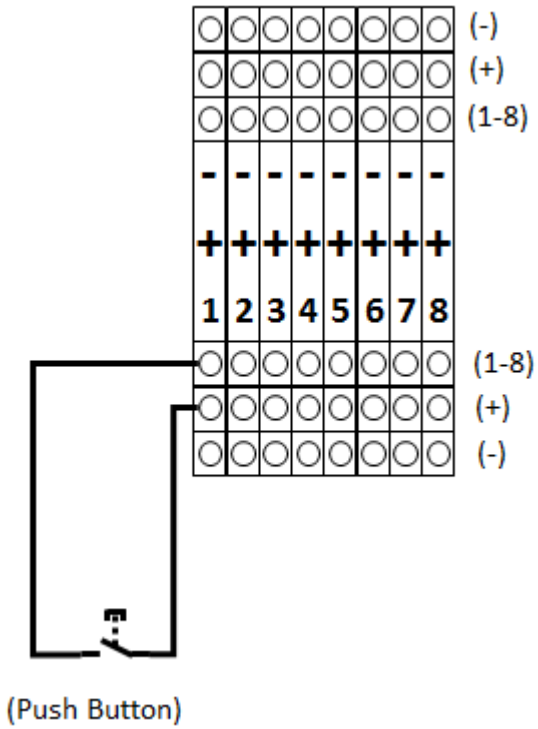
The inputs and outputs of the SIMATIC ET200 SP I/O module are wired to the terminal blocks on the right hand side of the kit. The left terminal block is used to connect digital inputs. The right terminal block is used to connect digital outputs.



Connecting a Digital Input

This section shows how to connect an additional digital input to the kit. A push button is used for demonstration purposes.

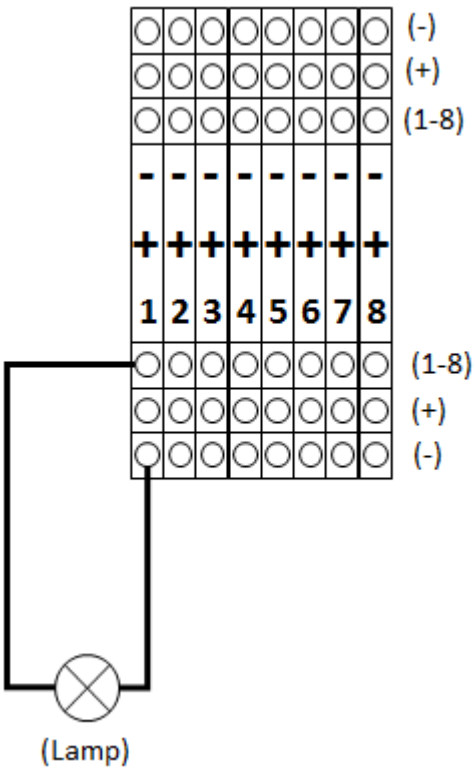
1. Connect the power supply of the button to the middle row of the I/O module (+24V).
2. Connect the input to the top row.



Connecting a Digital Output

This section shows how to connect an additional digital output to the kit. A lamp is used for demonstration purposes.

1. Connect the lamp to the top row of the I/O module.
2. Connect the common wire to the bottom row to close the electrical circuit.



After wiring the sensors or actuators to the inputs or outputs respectively switch the kit back on.

The next chapter describes how to assign variables to the inputs and outputs so that you are able to read data from newly connected sensors or control actuator functionality.

First Steps with CODESYS and the MFN 100

This chapter will give you an introduction on how to start working with the integrated soft PLC in the MFN 100. First, some configuration and installation of files and libraries are required.

NOTE

- Please download the CODESYS Development System V3 from store.codesys.com for this chapter.
  We recommend version 3.5 SP14 (32 bit) or newer.
- Please connect your workstation to the console port (P1) of the MFN 100.

Installing the Device Descriptions

After you have downloaded and installed the CODESYS Development System on your workstation, you have to install the device descriptions of the MFN 100 and the SIMATIC ET200 SP I/O module in the CODESYS Development System. The device descriptions have the following filenames:

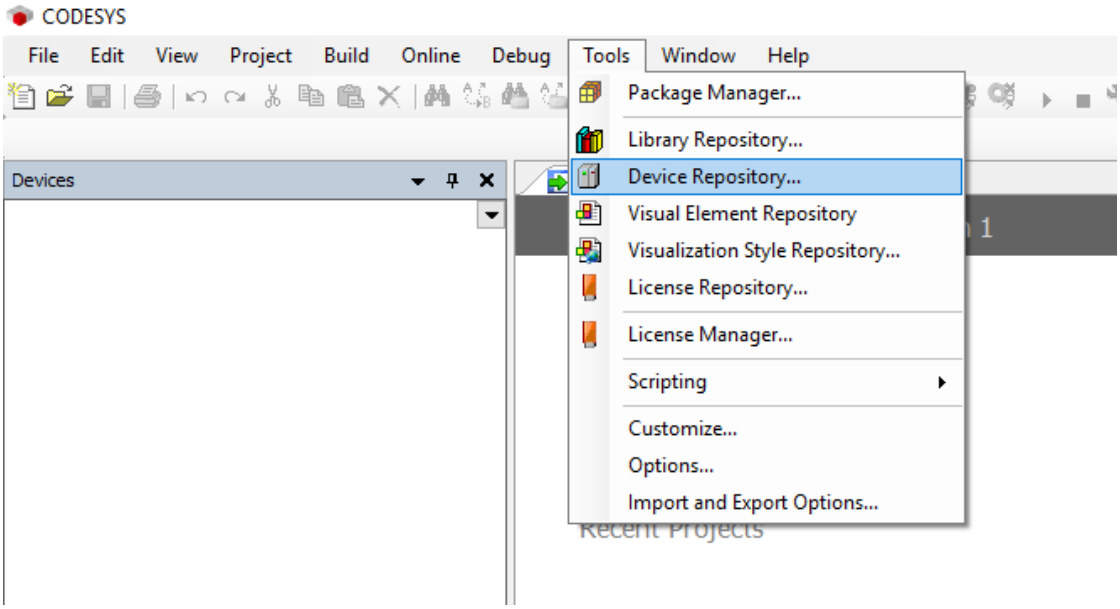
MFN 100	Nerve_MFN_100_V3.5.XX.X.devdesc.xml

SIMATIC ET200 SP	GSDML-V2.34-Siemens-ET200SP-20180926.xml

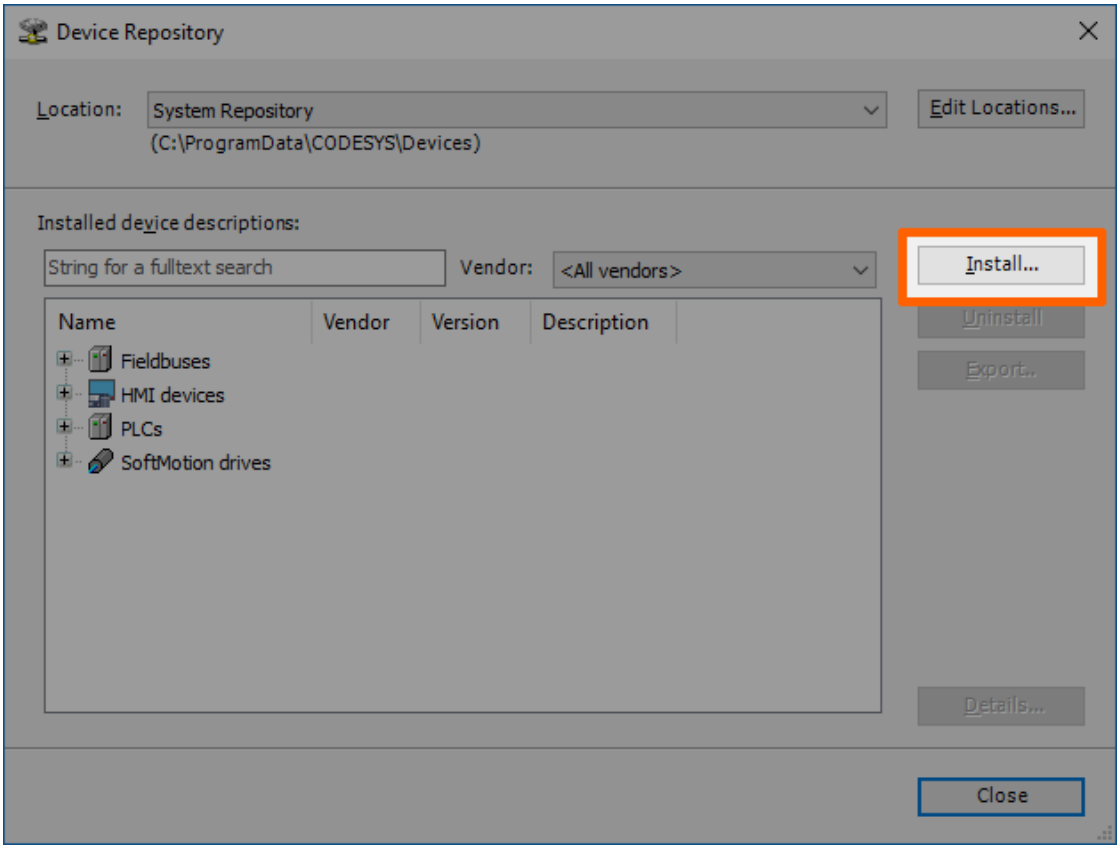
XX.X stands for the current version of the CODESYS Development System. The device descriptions of the MFN 100 and the SIMATIC ET200 SP I/O module have been sent as part of the delivery.

Please remember where you have saved the device description for the following steps.

1. Start the CODESYS Development System.
2. Go to **Tools > Device Repository**.



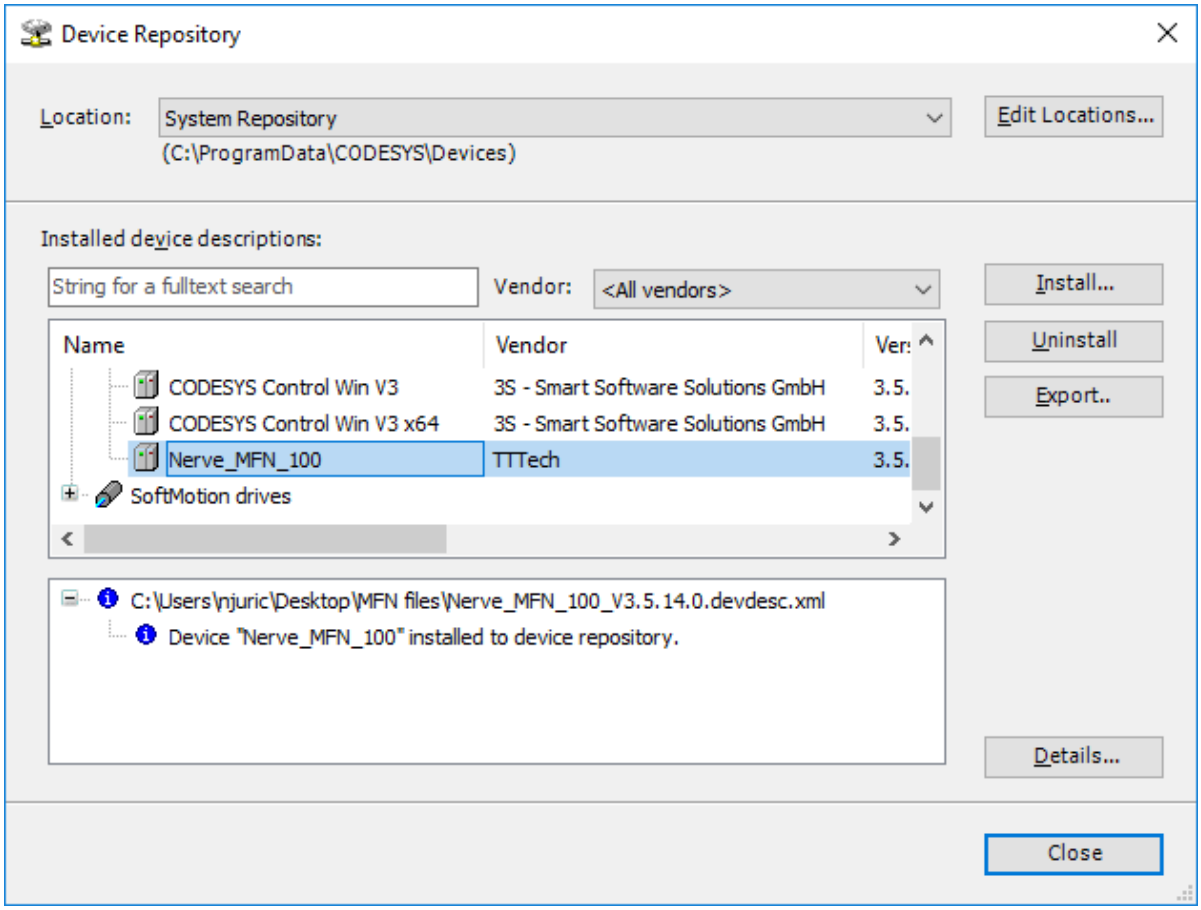
3. Click **Install**.



4. Go to the directory of the previously downloaded device description.

5. Select the device description of the MFN 100.
6. Click **Open**.
7. Repeat steps 3 to 6 to install the device description SIMATIC ET200 SP I/O module.

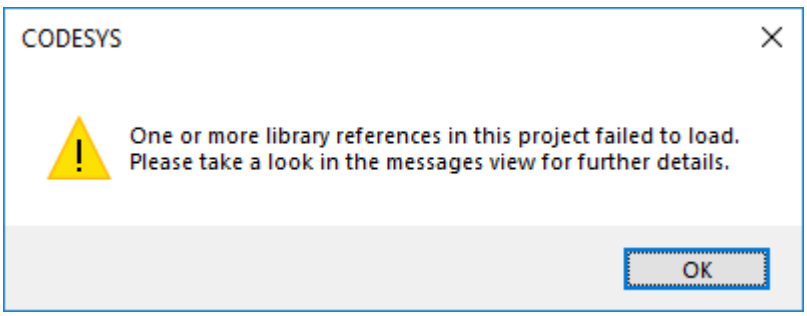
When the installation was successful, the MFN 100 and the SIMATIC ET200 SP I/O module will appear in the list of device descriptions in the middle of the window and you can close the window.



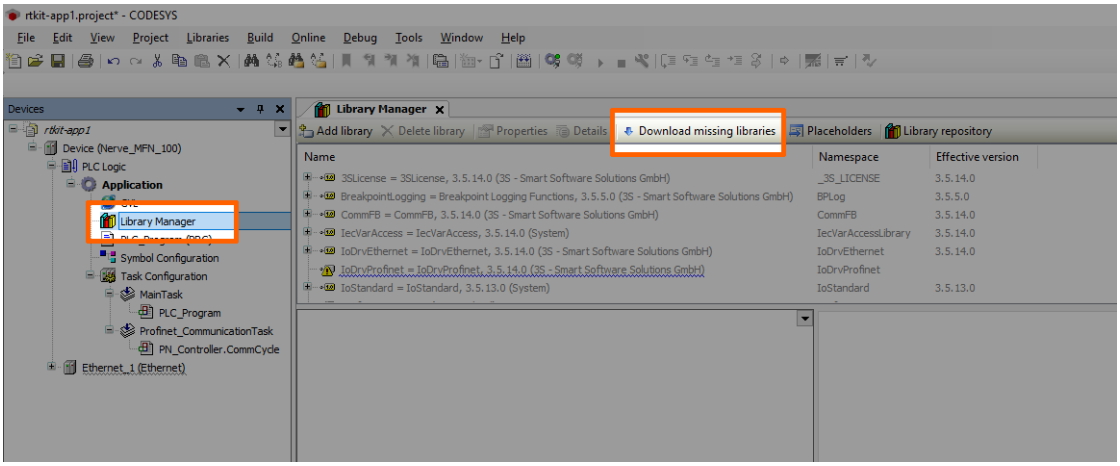
After installing the device description you can start working with the CODESYS Development System. However, libraries and device descriptions of generic devices might be missing so that the CODESYS Development System can work properly. The following chapters will walk you through the download process.

Downloading Missing Libraries

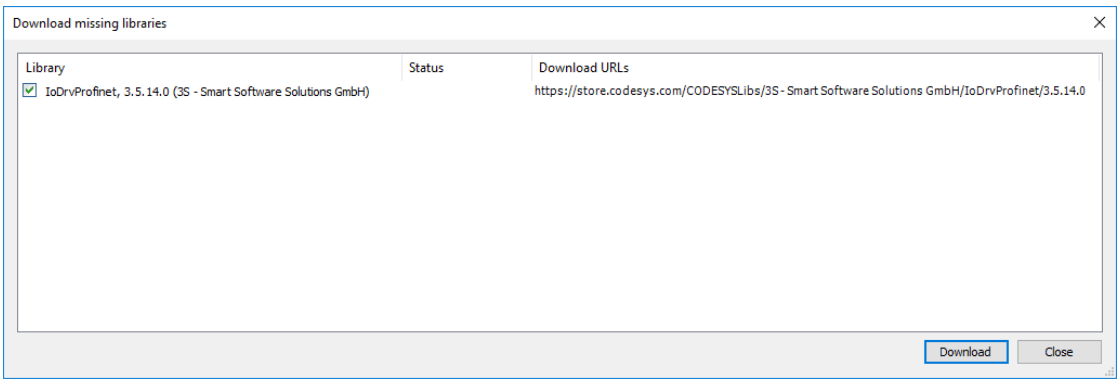
The error message for missing libraries might appear when you open or create a CODESYS project. The CODESYS Development System identifies the missing libraries automatically but you may have to repeat the following process a few times.



1. Open or create a CODESYS project.
2. If the error message about missing libraries appears, click **OK**.
3. Double-click **Library Manager** in the tree view on the left.
4. Click **Download missing libraries**.



5. Click **Download** in the new window.



6. Click **Close** when the download is finished.

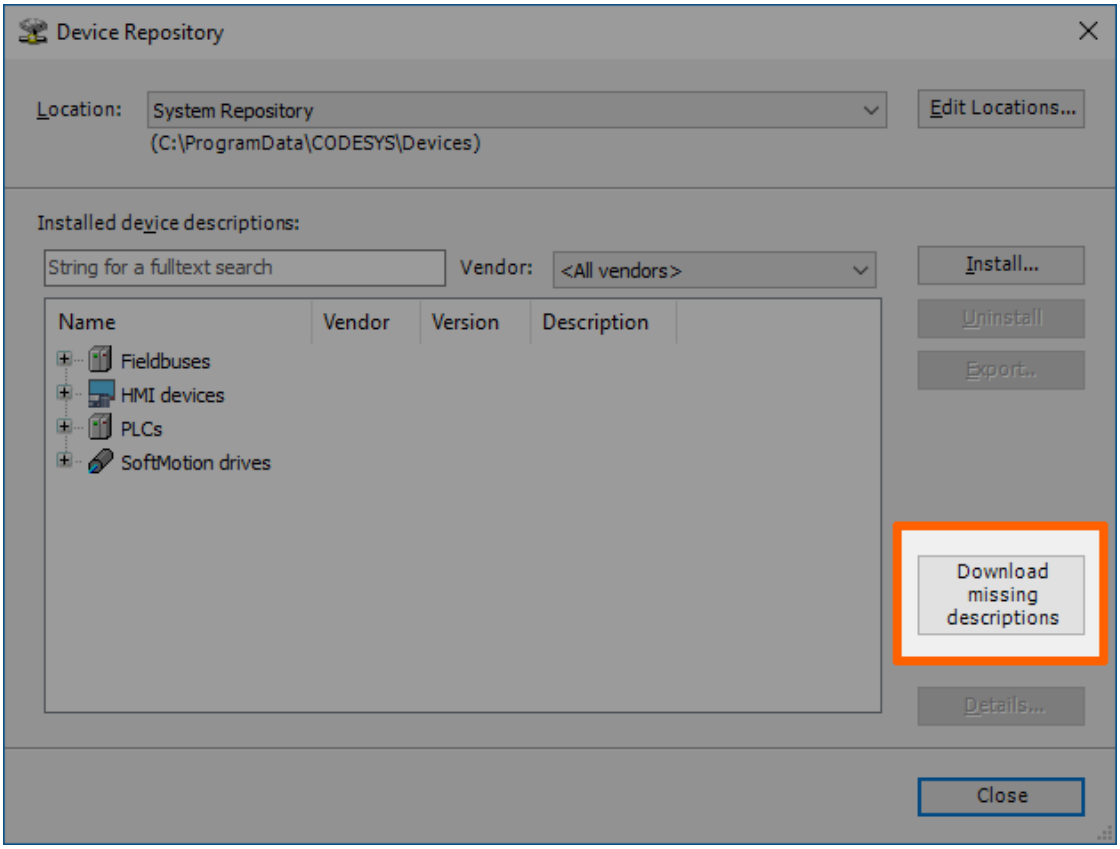
7. Repeat steps 3 to 5 until no more libraries appear in the download window.

Downloading Missing Device Descriptions

Apart from the device description for the MFN 100 that you have installed manually before, device descriptions of generic devices may be missing for the CODESYS Development System to function as intended. The CODESYS Development System will identify the missing device descriptions automatically but this time it will not generate an error message unless you try to load a CODESYS application into the MFN 100.

1. Click **Tools > Device Repository**.

2. Click **Download missing descriptions**.



NOTE

If no device descriptions of generic devices are missing, the button for downloading missing descriptions will not appear.

3. Click **Download** in the new window.

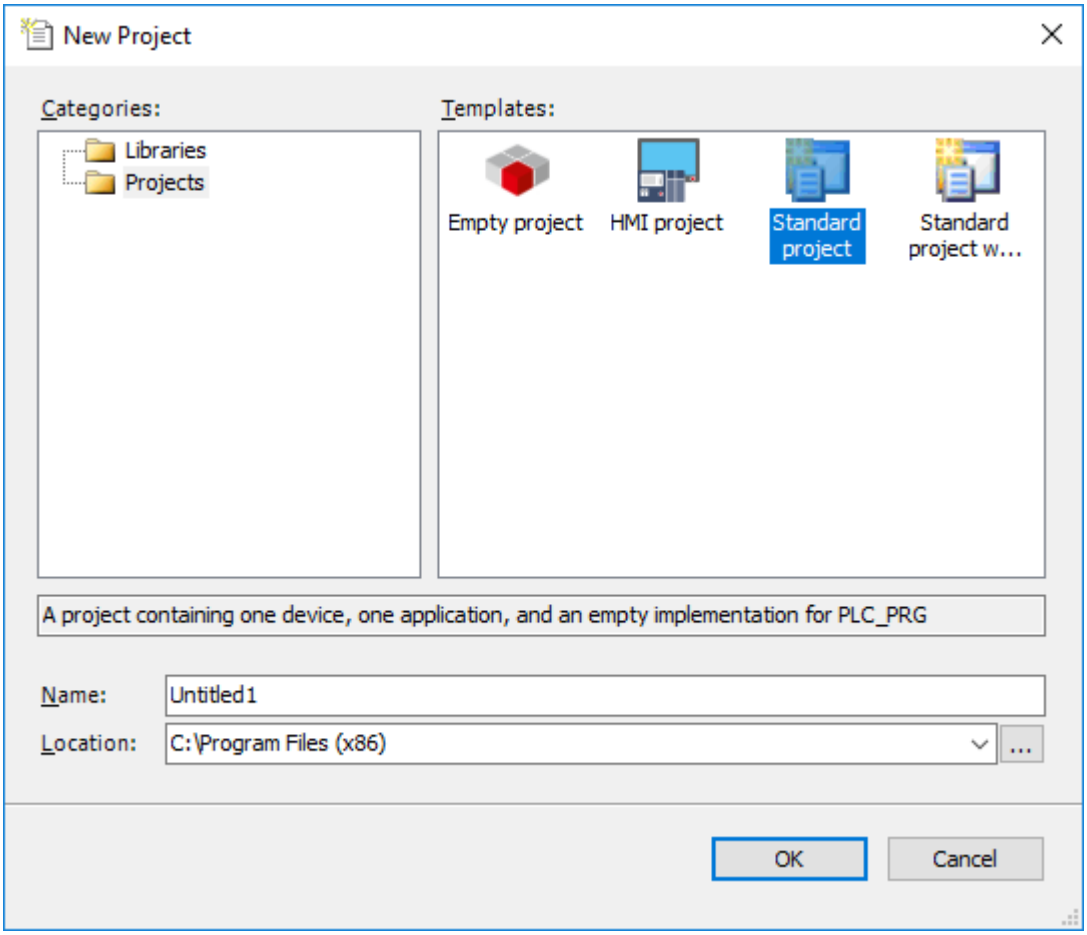
4. Click **Close** when the download is finished.

Creating a New CODESYS Project

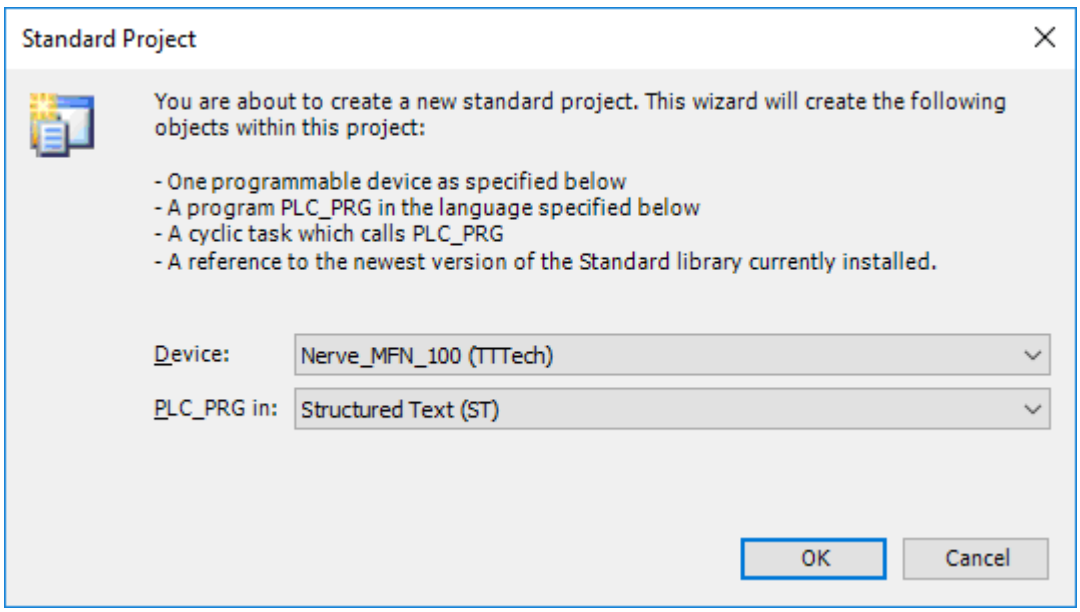
This example shows how to create a new project in the CODESYS Development System. The easiest way to get started is to create a **Standard project**.

1. Start CODESYS
2. Go to **File > New Project**.
3. Click **Standard project** on the right side among the templates.
4. Enter a name for the project.
5. Choose a **Location** where the project will be saved.

6. Click **OK** to save the project.

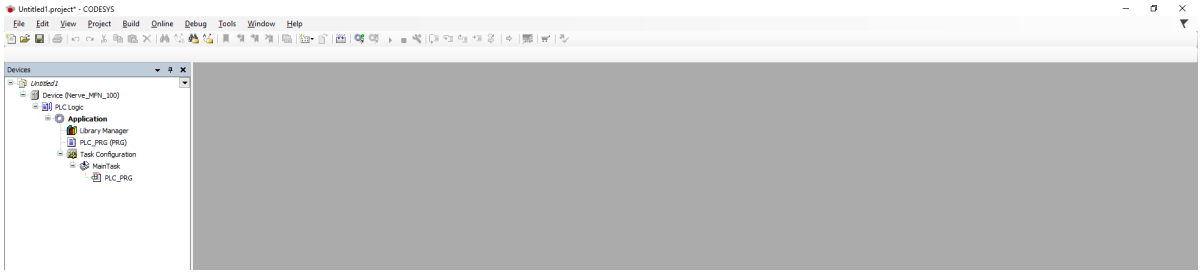


7. Select **Nerve_MFN_100 (TTTech)** as the device.



8. Click **OK**.

The result is an empty project that is open in the main view of CODESYS.



Working with the Default Applications

If you wish to work with existing applications first, you can modify the default applications app1.project and app2.project. They have been sent as part of the delivery.

1. Start CODESYS.
2. Go to **File > Open Project**.
3. Select the location where you have downloaded the default applications.
4. Select the application you want to work with.

Name	Date modified	Type	Size
app1.project	24.07.2019 14:55	CODESYS project	287 KB
app2.project	24.07.2019 14:57	CODESYS project	288 KB

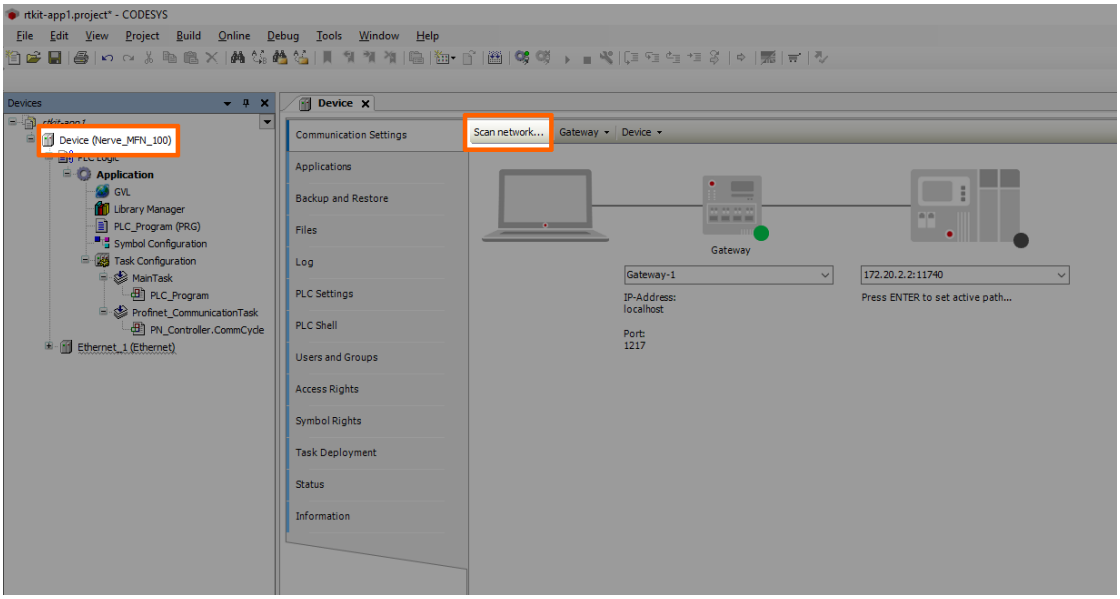
5. Click **Open**.

If you are opening the default applications for the first time, some libraries and device descriptions will be missing. Follow the instructions above to see how to download the missing files.

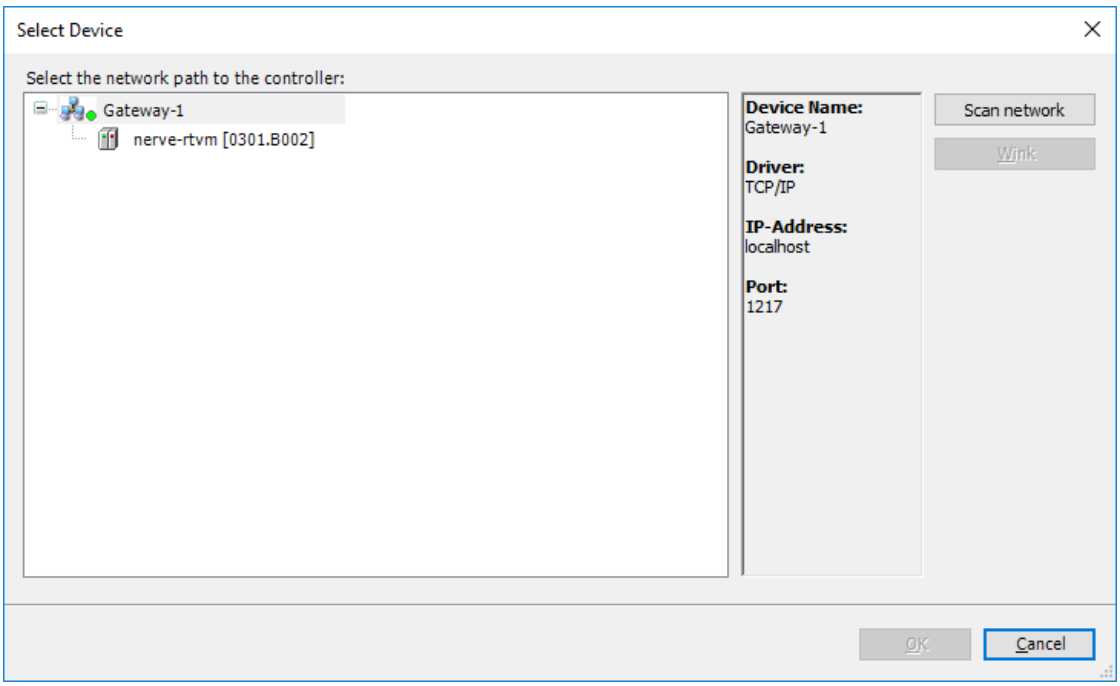
Connecting to the MFN 100

Before you can download CODESYS applications to the MFN 100, please make sure that the device description of the MFN 100 is installed in the CODESYS Development System.

1. Open or create a CODESYS project.
2. Double-click **Device (Nerve_MFN_100)** in the tree view on the left.
3. Go to **Communication Settings > Scan network...**.



4. Select the MFN 100 (here **nerve-rtvm [XXXX.XXXX]**) in this window.



NOTE

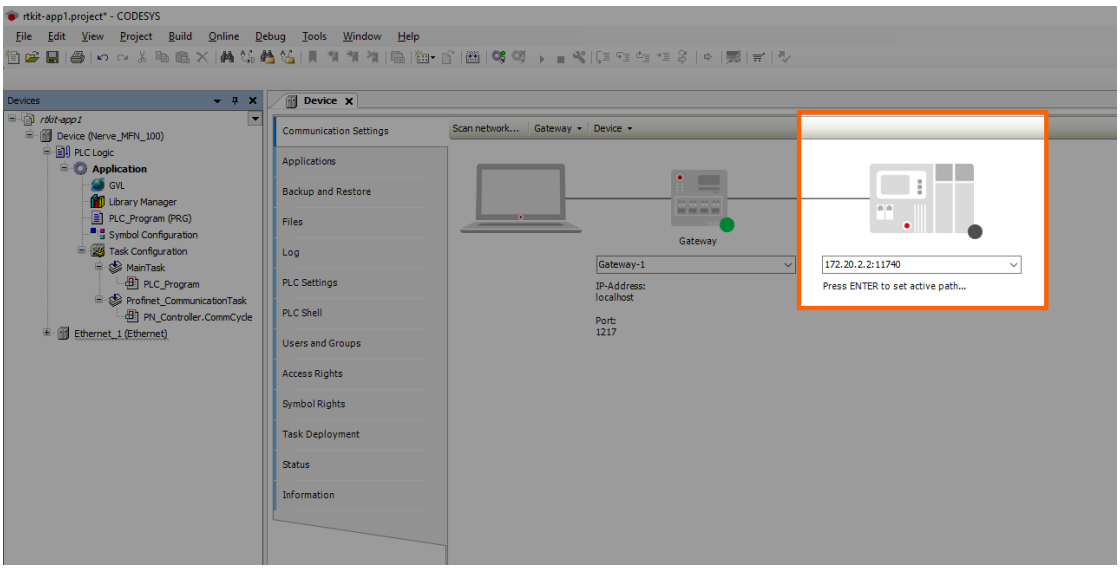
When more than one network is active on your workstation, it sometimes happens that the MFN 100 cannot be found. Please continue reading if the MFN 100 does not appear in this window.

5. Click **OK**.

Typically the MFN 100 will be found automatically. If the MFN 100 cannot be found, you have to enter the IP address and port of the CODESYS runtime manually.

1. Double-click **Device (Nerve_MFN_100)** in the tree view on the left.
2. Go to **Communication Settings** in the middle of the window.

3. Enter 172.20.2.2:11740 in the text box under the device on the right.



4. Press Enter.

The CODESYS Development System is now connected to the MFN 100 and you can download applications into the CODESYS runtime.

Downloading an Application to the MFN 100

You can load CODESYS applications directly into the MFN 100. However, before you can download an application into the MFN 100 it needs to be free of errors.

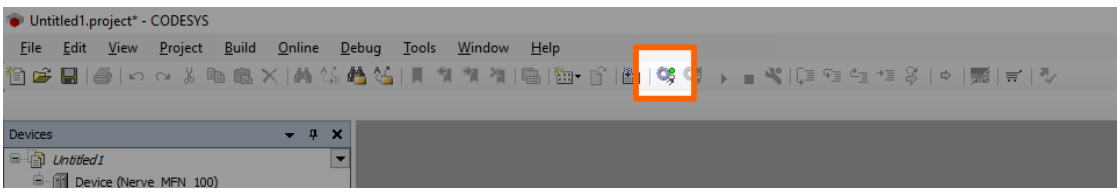
The process of downloading an application is slightly different if you are downloading an entirely new application into the MFN 100 or if you are updating an application that has already been downloaded into the MFN 100. If you are updating an application that you have downloaded to the MFN 100 before, please continue with Downloading an Updated Application to the MFN 100 further down below.

Downloading a New Application to the MFN 100

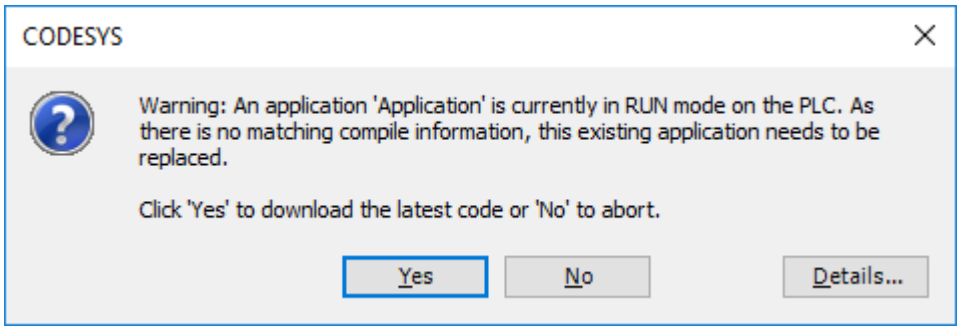
Once you have created a project and you are finished with programming, you can download the CODESYS application to the MFN 100 directly.

1. Open the CODESYS project you want to load into the MFN 100.

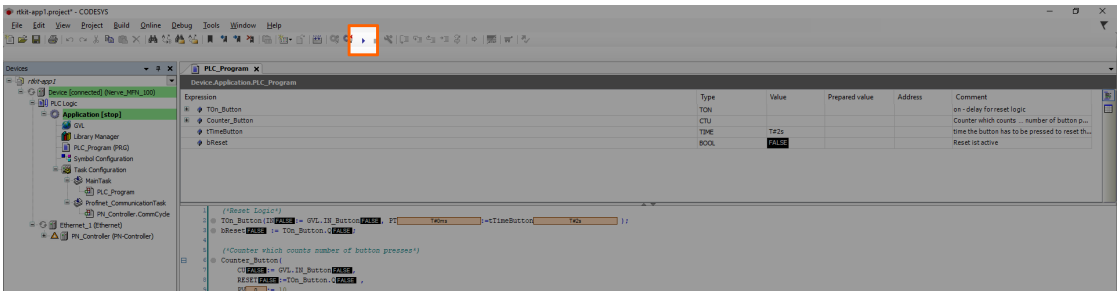
2. Click the **Login** symbol in the CODESYS menu bar.



3. Click **Yes** in the pop-up window.



4. The application is stopped now. Click the **Play** symbol in the CODESYS menu bar.



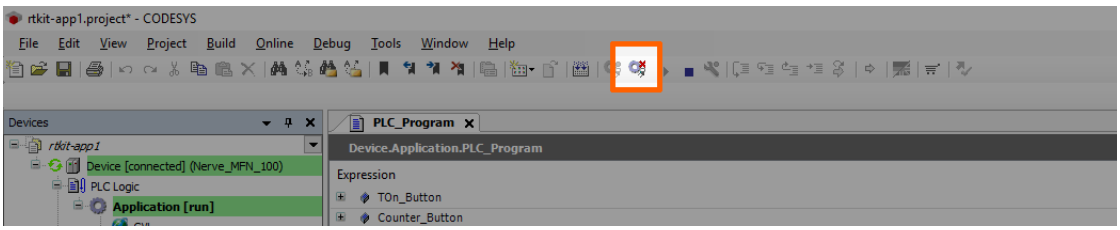
The application is now loaded to the MFN 100.

Downloading an Updated Application to the MFN 100

If you update an application after you have loaded it into the MFN 100, you need to download it into the MFN 100 again. The download process is slightly different from downloading a new application into the MFN 100.

1. Stop the CODESYS application that you have loaded into the MFN 100 through the local UI.

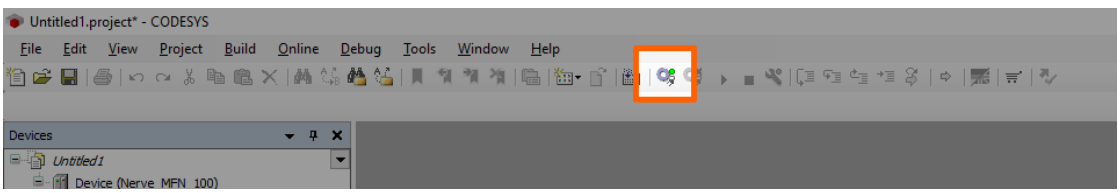
2. Click the **Logout** button in the CODESYS toolbar.



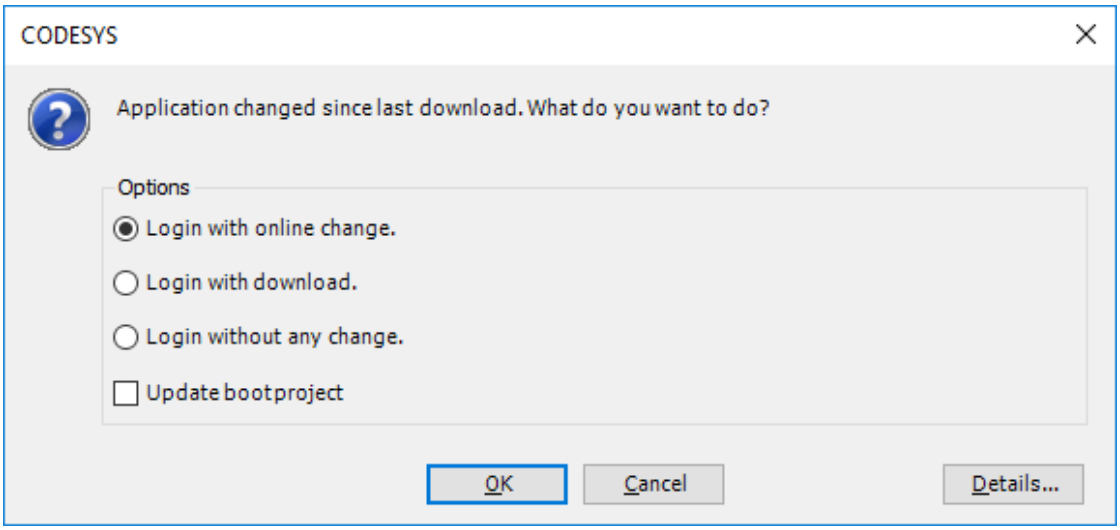
3. Expand **Device (Nerve_MFN_100) > PLC Logic > Application**.

4. Double-click **PLC_Program (PRG)**.
5. Perform your changes.

6. Click the **Login symbol** in the CODESYS menu bar.



7. In the pop-up window, select one of the options.



Item	Description
Login with online change.	The updated application will be loaded into the MFN 100. Variable values will not be reset. If the application was running before, it will be running after the download.
Login with download.	The updated application will be loaded into the MFN 100. Variable values will be reset. The application is stopped.
Login without any change.	The updated application will not be loaded into the MFN 100 but the code will keep your changes.

8. Click **OK**.

The application is now loaded to the MFN 100.

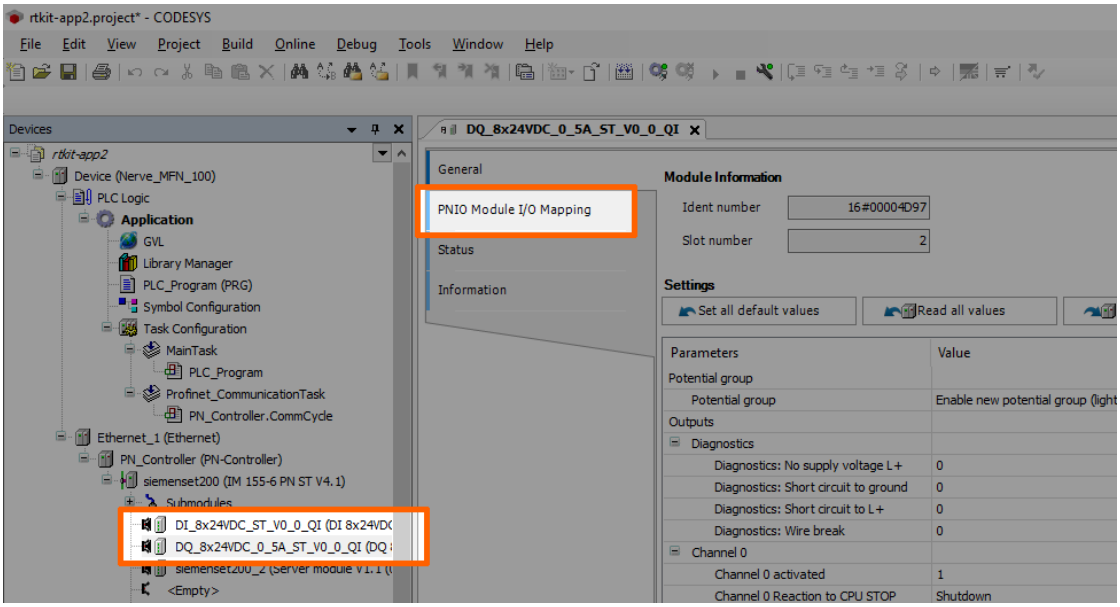
NOTE

For more help with programming PLC applications in the CODESYS Development System go to help.codesys.com.

Allocating Variables to Inputs or Outputs

After connecting new sensors and actuators, you need to assign variables to the I/O channel in CODESYS.

1. Open a CODESYS project.
2. Expand **Device (Nerve_MFN_100) > PLC Logic > Ethernet_1 > PN_Controller > siemenset200 (IM 155-6 PN ST V4.1)** in the tree structure on the left.
3. Double-click **DI_8x24VDC_ST_V0_0_QI (...)** for digital inputs.
   Double-click **DQ_8x24VDC_0_5A_ST_V0_0_QI (...)** for digital outputs.
4. Select **PNIO Module I/O Mapping**.

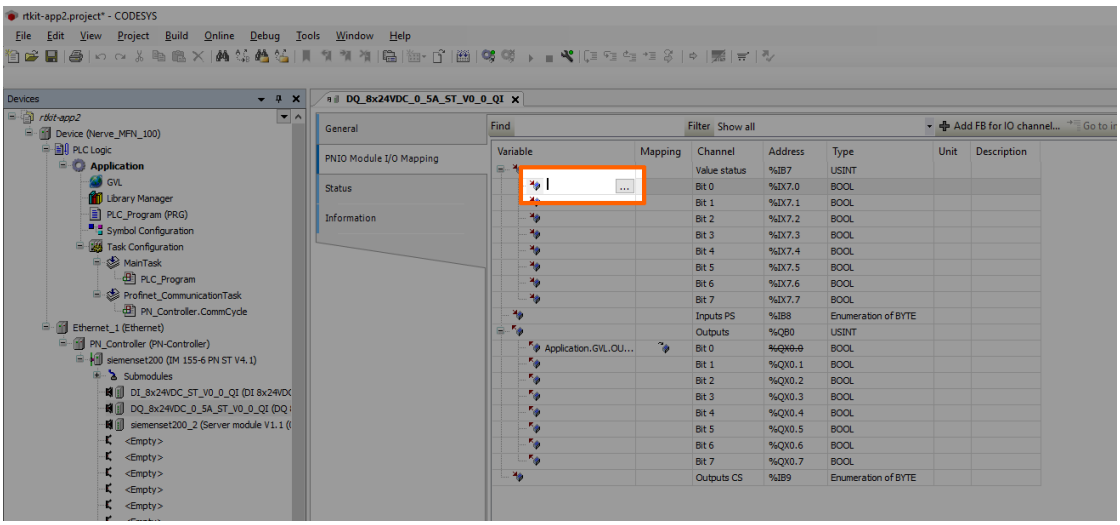


5. Fully expand the tree view.

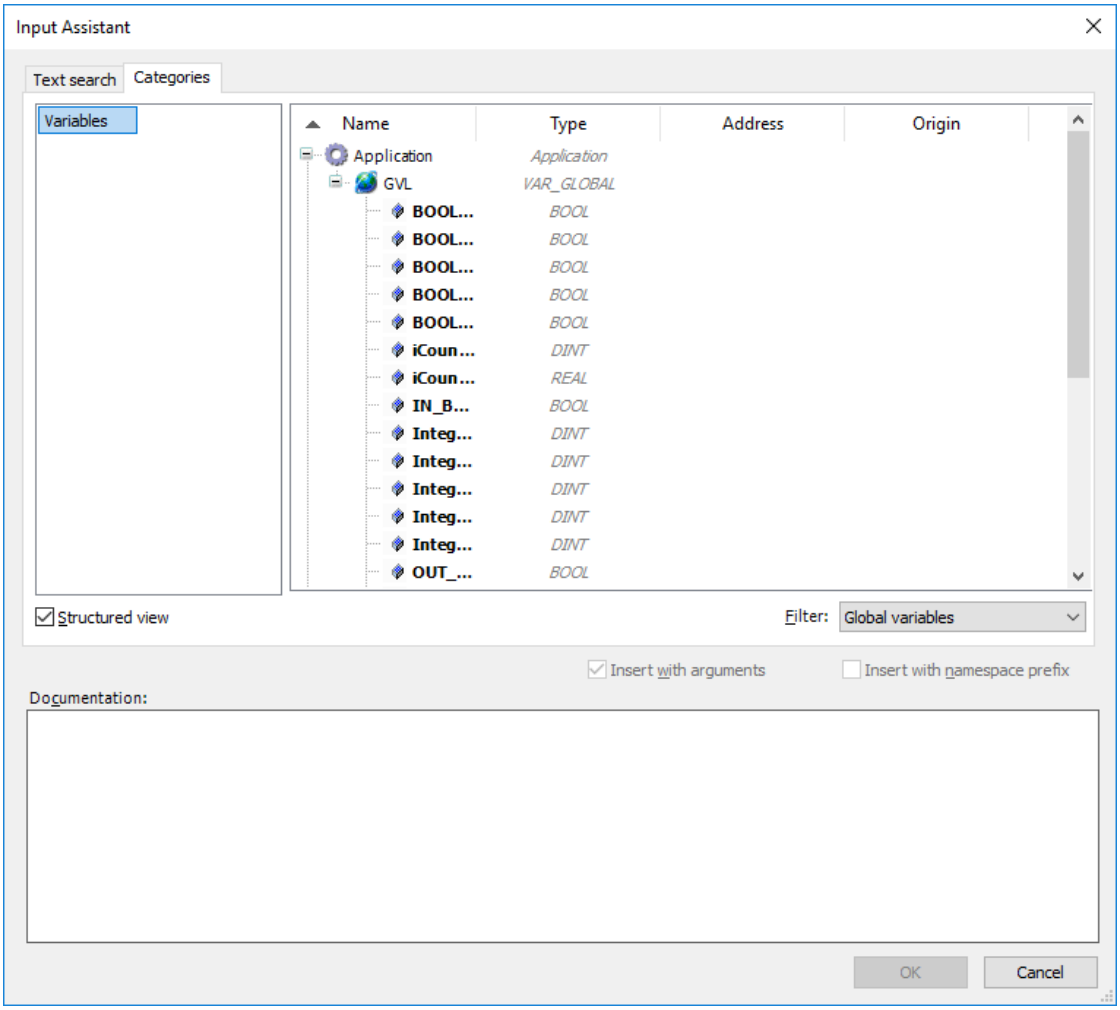
6. Double-click the variable slot you would like to assign.

> **NOTE**
>
> The inputs in this view do not match the physical inputs of the I/O module on the kit. The inputs here go from 0 to 7. The physical inputs go from 1 to 8. Therefore input 0 in this view represents the physical input 1 on the I/O module. This also applies to outputs.

7. Click the three dots next to the variable slot.



8. Expand **Application > GVL** in the new window.



9. Select the variable you would like to assign.

> **NOTE**
>
> Please make sure to select a variable of the same type as the input, i.e., a **BOOL** variable for a **BOOL** input or output.

10. Click **OK**.

You can use the assigned variables to read data from connected sensors or to control actuator functionality.

> **NOTE**
>
> For more help with programming PLC applications in the CODESYS Development System go to help.codesys.com.

Data Transfer from CODESYS to the Management System

NOTE

The configuration and visualization of new variables with the Nerve Datapath and Grafana is not available in version 2.0 of Nerve Blue. This chapter covers steps and information before the the visualization is configured. The Nerve Datapath and Grafana visualization will be available with version 2.1.

The kit sends data that is generated in CODESYS from the MFN 100 to the Management System using variables. This chapter gives a quick overview on how to configure new variables.

Overview

The CODESYS runtime has an integrated OPC UA server which you can configure via the CODESYS Development System. The data from the OPC UA server is received by an OPC UA client and written into the database in the Management System.

You can use the following global variables to transmit data to the Management System:

Global Variables			
BOOL_1	Integer_1	Real_1	STRING_1
BOOL_2	Integer_2	Real_2	STRING_2
BOOL_3	Integer_3	Real_3	STRING_3
BOOL_4	Integer_4	Real_4	STRING_4
BOOL_5	Integer_5	Real_5	STRING_5

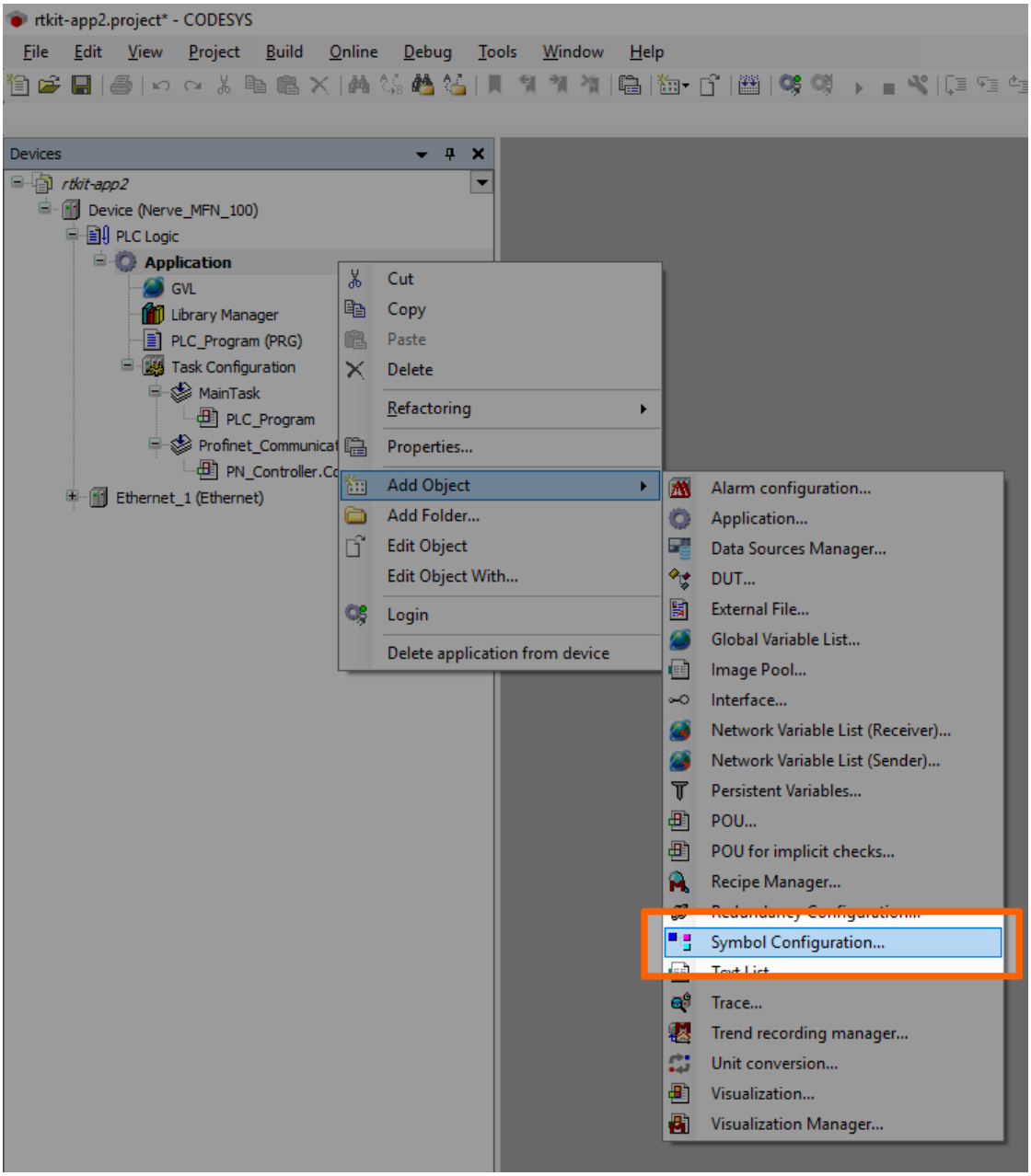
The following instructions are only needed if new variables are added or a completely new CODESYS project is created.

Configuring the CODESYS OPC UA Server

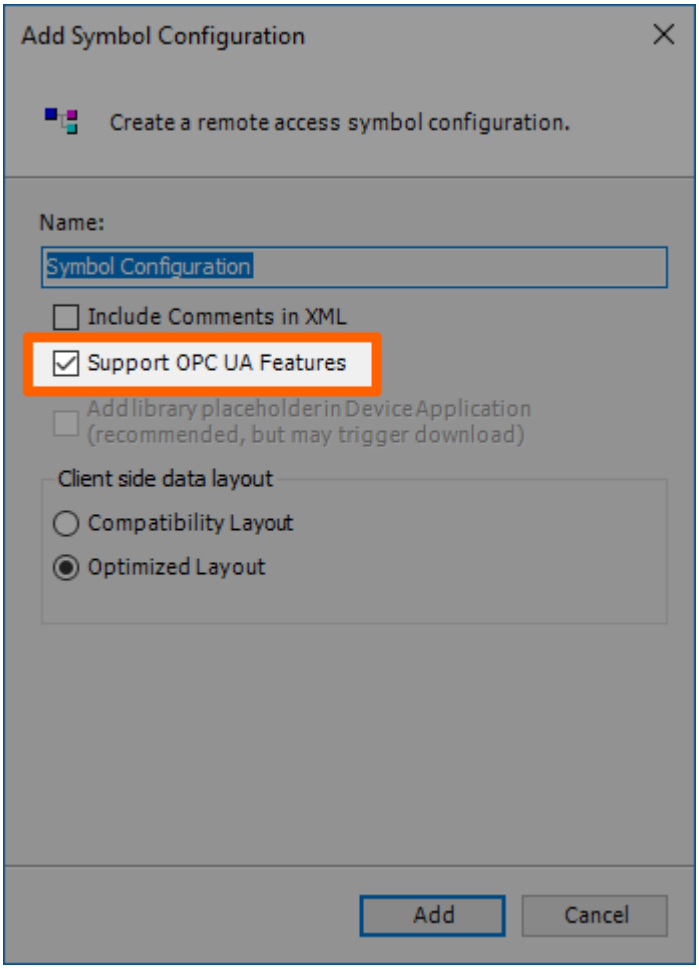
In the CODESYS Development System you can configure which variables are available in the OPC UA server. Configuring variables requires an object called **Symbol Configuration**.

1. Open the CODESYS project you want to configure the variables in.
2. Expand **Device (Nerve_MFN_100) > PLC Logic**.
3. Right click **Application**.
4. Select **Add Object**.

5. Click **Symbol Configuration...**



6. Tick **Support OPC UA Features**.



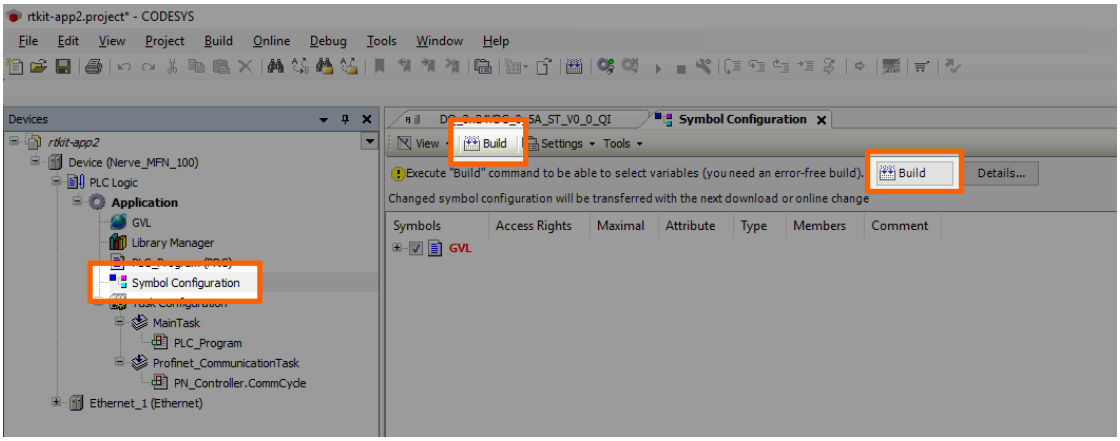
7. Click **Add**.

Adding Variables to the OPC UA Server

Before you can add variables to the OPC UA server, you have to add the object Symbol Configuration to the tree structure. Please see the instructions above on how to add the Symbol Configuration to the tree structure.

1. Open the CODESYS project you want to configure the variables in.
2. Expand **Device (Nerve_MFN_100) > PLC Logic > Application**.
3. Double-click **Symbol Configuration**.

4. Click **Build**.



5. Expand **GVL**.

6. Tick the variables you want to add to the OPC UA server.