Install or update
SOLIDWORKS 2023

cadmes
a Bechtle group company

Introduction

This manual explains step by step how to install or update SOLIDWORKS. Should you have any issue or question, please contact our support team: support@cadmes.com or +32 71 35 14 83.

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1 | Check prerequisites

If you have SOLIDWORKS PDM, please contact us. We need to update the PDM server and SOLIDWORKS at the same time.
Check that you are under subscription (contact us if you are not sure).

1   Check the compatibility with the new version:
    » Software
    » Graphic card

2   Check that collaborators (other employees, customers, partners) are using the same version to be able to share files easily.

3   If you install your license on a new computer, get your SOLIDWORKS Serial Number with you.
    » You should have received it by email.
    » Otherwise, contact us.

4   Check your license type with the help of the serial number: 9000 0053 1234 5678 ABC9 DEFH.
    » If the 3rd number is 1, you have a network license.
    » If the 3rd number is 0, you have a standalone license.

5   If you have a network license, check the compatibility with your license server:
    » If it is a virtual server, virtualization system can be found here.
    » In all cases, the operating systems supported are the same than SOLIDWORKS plus Windows Server found here under "Server Products".

	SOLIDWORKS 2021 (EDU 2021-2022)	SOLIDWORKS 2022 (EDU 2022-2023)	SOLIDWORKS 2023 (EDU 2023-2024)
Client Products (e.g. SOLIDWORKS)			
Windows 11, 64-bit	✕	✓ (SW2022 SP2)	✓
Windows 10, 64-bit	✓	✓	✓
Server Products (e.g. SOLIDWORKS PDM, SNL Manager)			
Windows Server 2022	✕	✓ (SW2022 SP4)	✓
Windows Server 2019	✓	✓	✓
Windows Server 2016	✓	✓	✓ (End of Life: SW 2023)

6   Finally, SOLIDWORKS gives a lot of information and a complete guide for installing and administering:
    https://my.solidworks.com/support/adminguides

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2 | Short version

Our [YouTube video](#) (in French) can guide you through the installation and update of SOLIDWORKS.

1	If you just bought SOLIDWORKS with a standalone license (third number of the serial number is 0):
	If not already done, create your SOLIDWORKS/3DEXPERIENCE account [here](#).

2   Download SOLIDWORKS installation manager [here](#) (login with your account).

3   Start the setup.exe file and follow the instructions, choosing "Install on this computer".
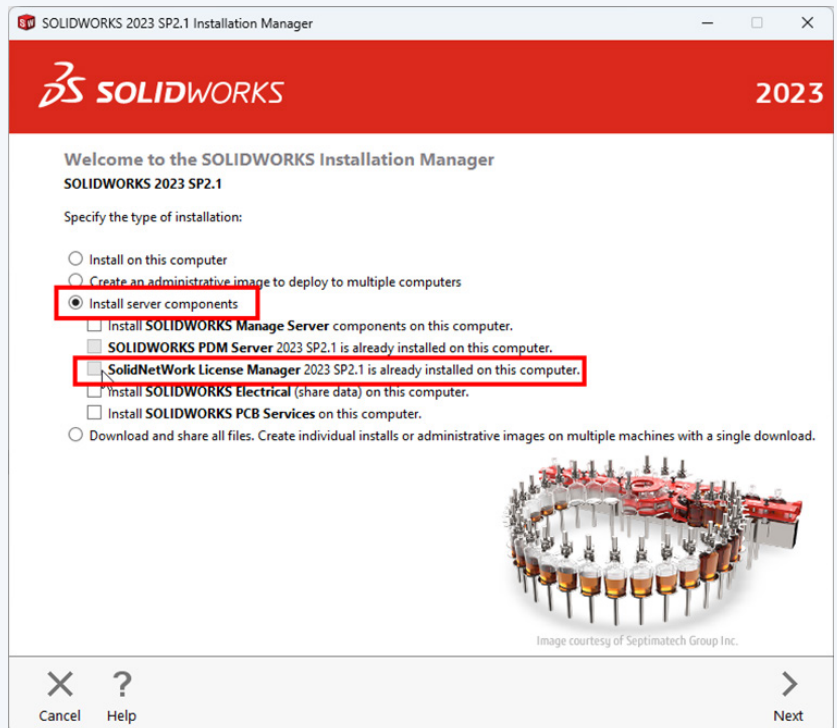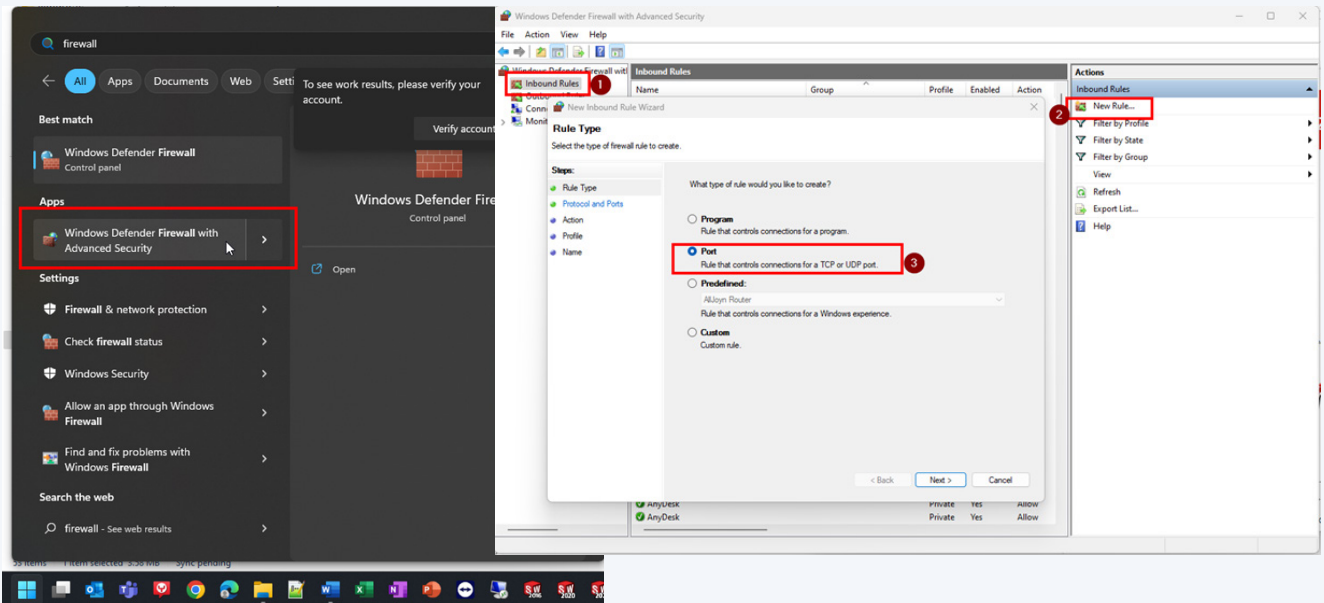
2 | Short version

4   If you just bought SOLIDWORKS with a network license (third number of the serial number is 1):

If not already done, create your SOLIDWORKS/3DEXPERIENCE account here. Download SOLIDWORKS installation manager here (login with your account).

On your license server (should be always accessible from all users), start the setup.exe file and choose "Install server components" and "SolidNetWork License Manager".

5   Follow the instructions.

6   If you have a firewall on your server (check the one from Windows), allow the communication through port 25734 and 25735 for TCP and UDP.
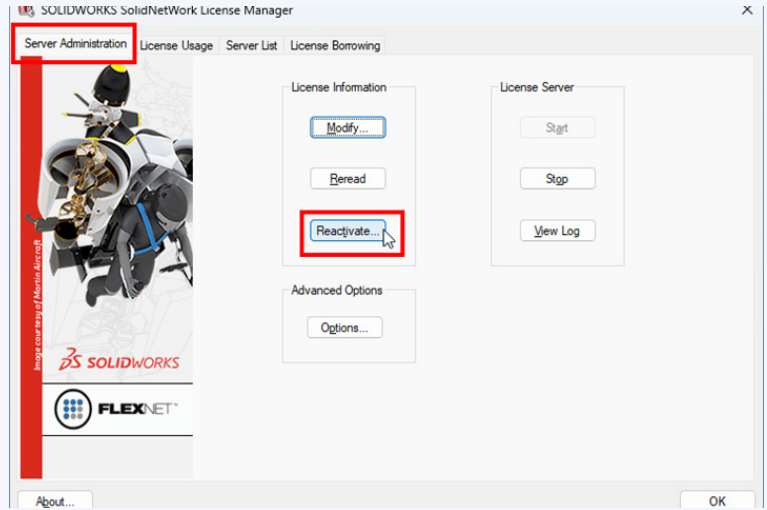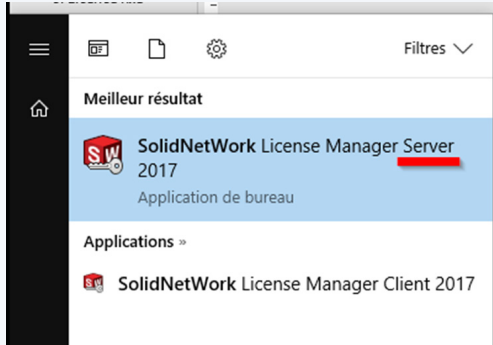

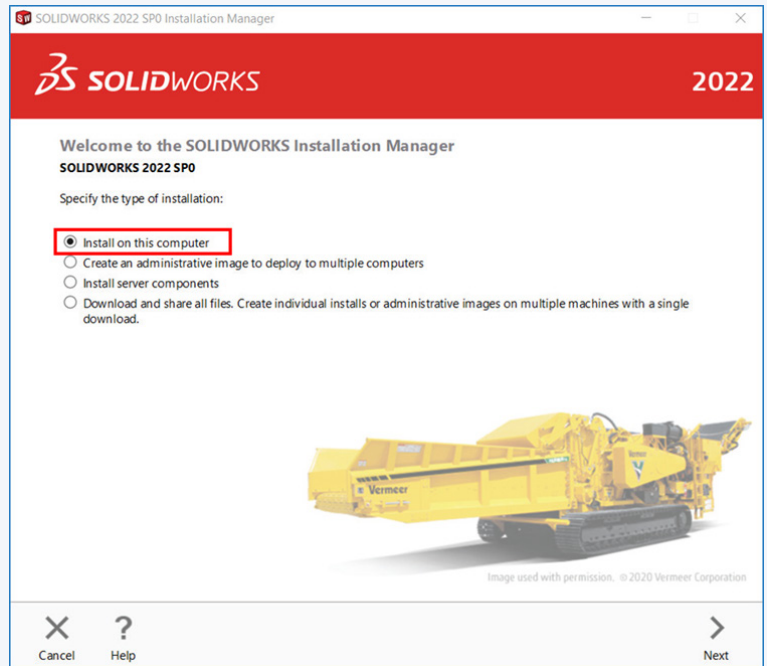**7**   Start the application SolidNetWork License Manager Server.

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2 | Short version

8   You should be asked to activate your licenses, otherwise click on "Reactivate".

9   On the client computer, start the same setup.exe file, choose "Install on this computer", and follow the instructions:

10   When asked for the license server, enter "25734@ServerName" (replacing "ServerName" with the name of your license server).

Follow the detailed version below in case of difficulties or call us.

3 | You already have SOLIDWORKS (you update your version)

Start with saving your SOLIDWORKS settings.

1   Create a zip backup for each setting.

2   Gather all your customized file and settings in a specific folder.

3   Save your SOLIDWORKS settings: name the file with the current version of SOLIDWORKS, the date of today and the username. Place the file in your specific folder for your customized SOLIDWORKS files and settings.

Be aware that this doesn't save your templates, material database and other customized files.
It only saves their location information:
where the files are but not the files themselves.
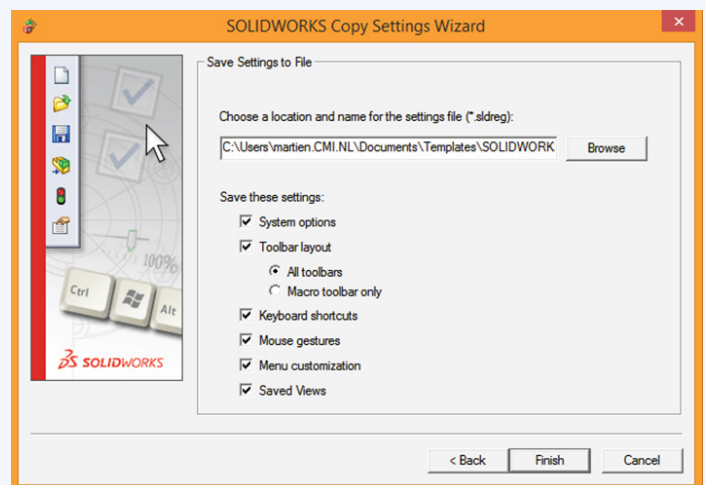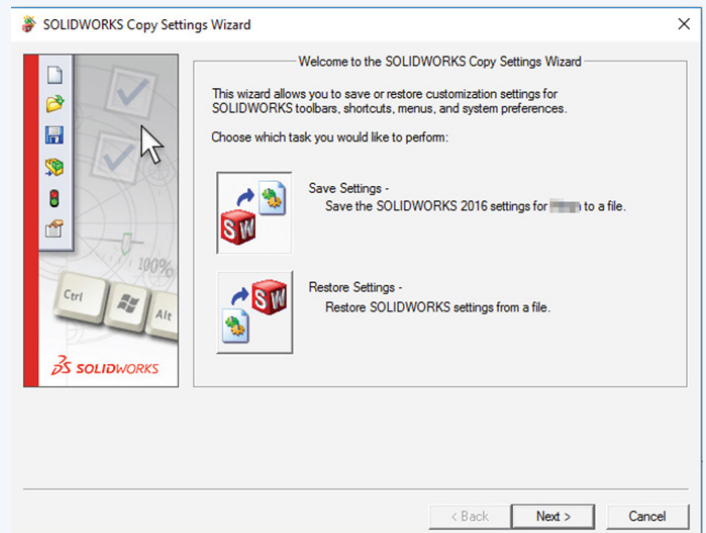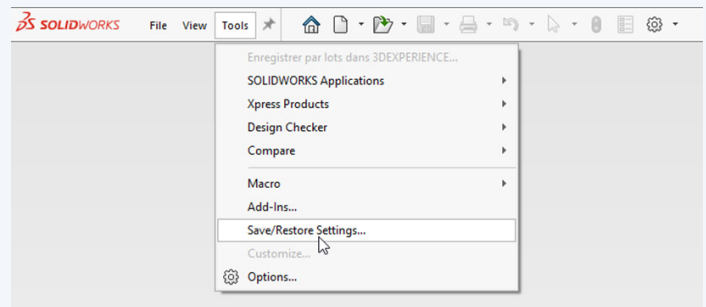
3 | You already have SOLIDWORKS (you update your version)

4 Also save as backup all your customized files:
» Templates, sheetformats, tables, material databases, Etc.
» See in "Options", "File location": if you have correctly set up your SOLIDWORKS.
All your customizations should be outside default locations:
  » C:\Program Files\SOLIDWORKS Corp\ SOLIDWORKS\Lang\english
  » C:\ProgramData\SolidWorks\SOLIDWORKS 2023\ lang\ english
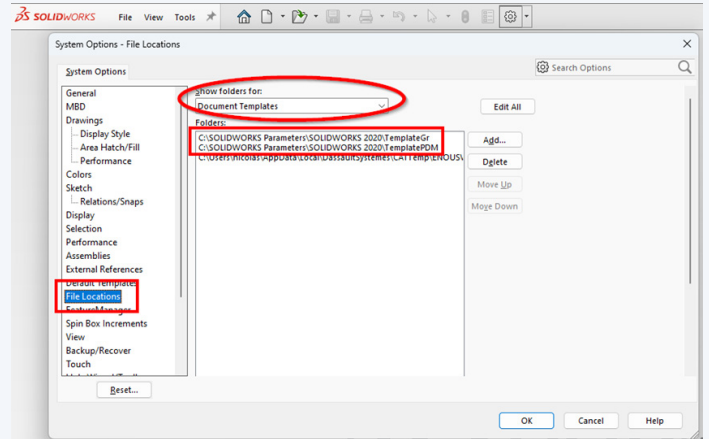
5 If your toolbox is customized (if not sure, do it), save as backup the folder (create a zip with the current version in the name):
  » To know where it is, go into SOLIDWORKS, "Options", "Hole Wizard/Toolbox":
  » By default, it is here: C:\SOLIDWORKS Data
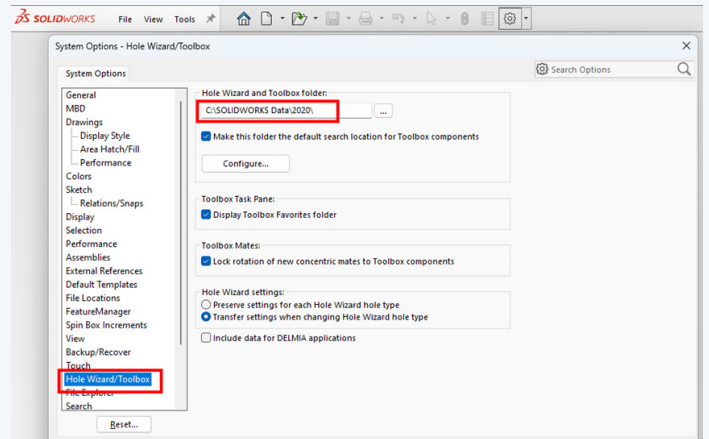
3 | You already have SOLIDWORKS (you update your version)

Download the installation files

1   If not already done, create your SOLIDWORKS/3DEXPERIENCE account here.

2   Download SOLIDWORKS installation manager here (login with your account).
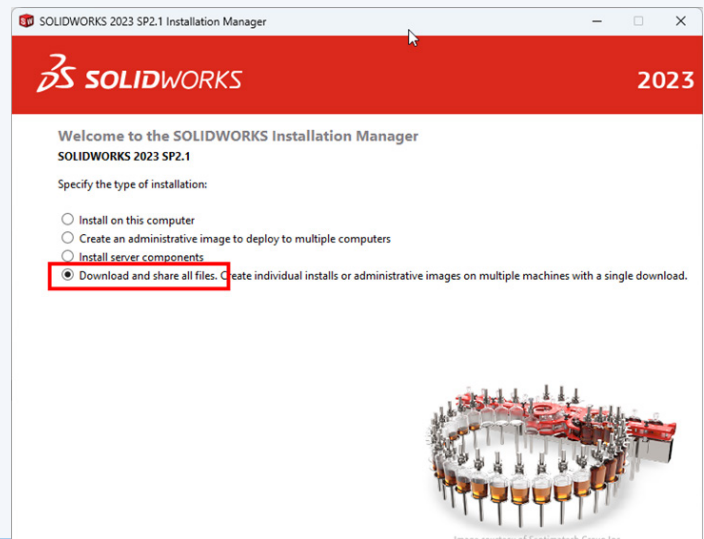

3   This is only the installation manager.
Be sure to have all the installation files and share them with your colleagues on your network:
  » Start the setup.exe downloaded above.
  » Choose "Download and share all files", follow the instructions.

# 3 | You already have SOLIDWORKS (you update your version)

Uninstall your current SOLIDWORKS version

1 Several methods can be used to update SOLIDWORKS:
» Direct update: it usually works but we prefer to uninstall before to clean up the computer.
» Install the new version in parallel with the old one: not advised because of reliability and official support from SOLIDWORKS.
» Uninstall the old one and install the new one: our recommendation.

Do not worry: when we uninstall SOLIDWORKS, we still keep all our settings, customizatiosn, serial numbers.
And we just save them as backup (did you ?).

2 To uninstall SOLIDWORKS:
Right click on the start menu in Windows,
application (at the top).

3 Find SOLIDWORKS, select it and uninstall it.
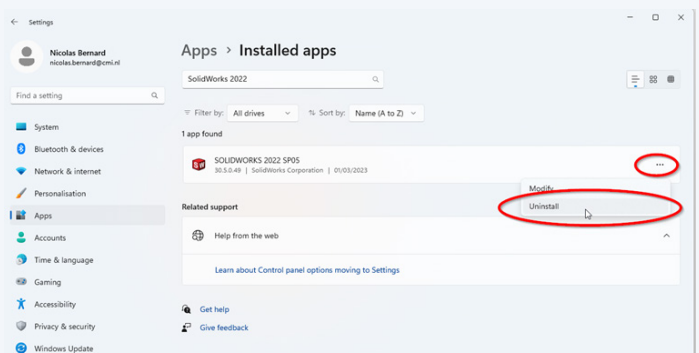
4 Follow the instructions with the default options.
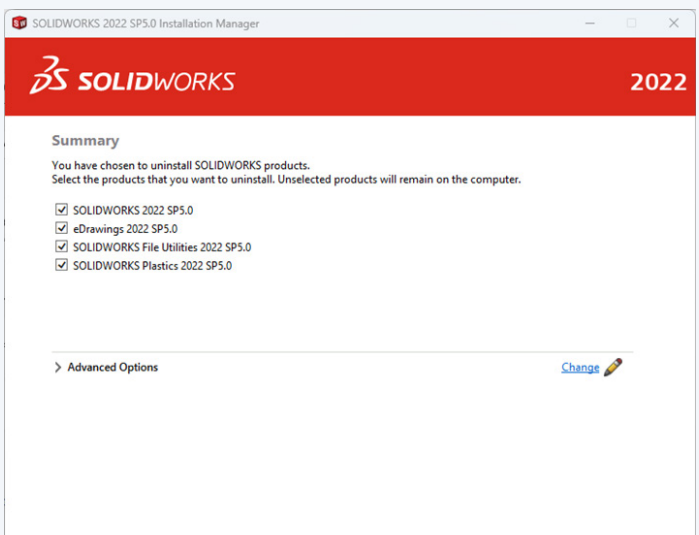
5 If you did not deactivate your license, you can do it now (you will activate them again after)

Once it is uninstalled, go to the next section depending on your license type (network or standalone).

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4 | You're new to SOLIDWORKS

If you haven't created a SOLIDWORKS/3DEXPERIENCE account yet, please click here and download SOLIDWORKS installation manager here.

Depending on your license type, go to the right chapter:
» If the third number of your serial number is 1 you are using a **Network license › go to page 12**
» If the thrid number of your serial number is 0 you are using a **Standalone license › go to page 14**
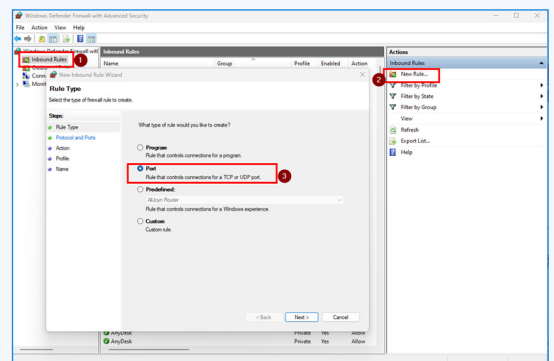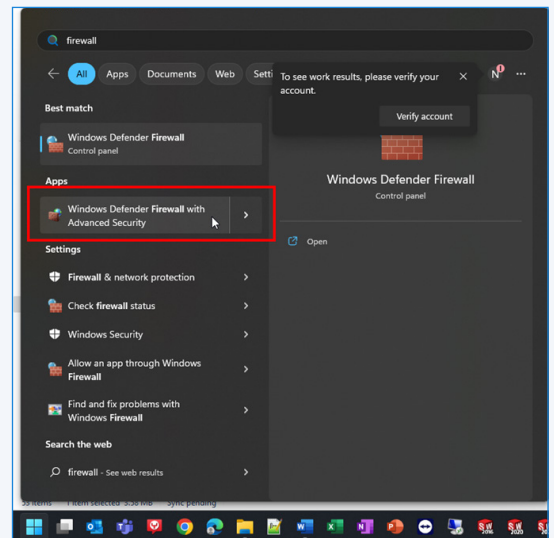
5 | Network license installation

1 Install License Manager on your server (should be accessible by all users):
» On the server, right click the setup.exe file you have just downloaded and start as administrator.
» Choose "Install server components" and "SolidNetWork License Manager".
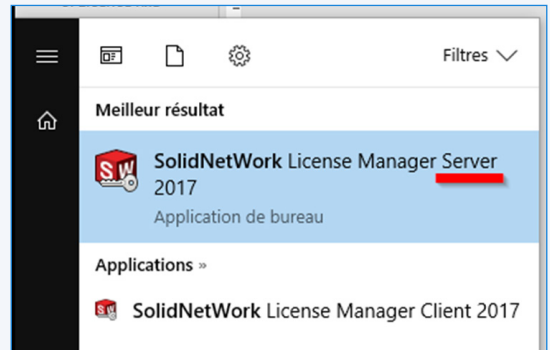» Follow the instructions.
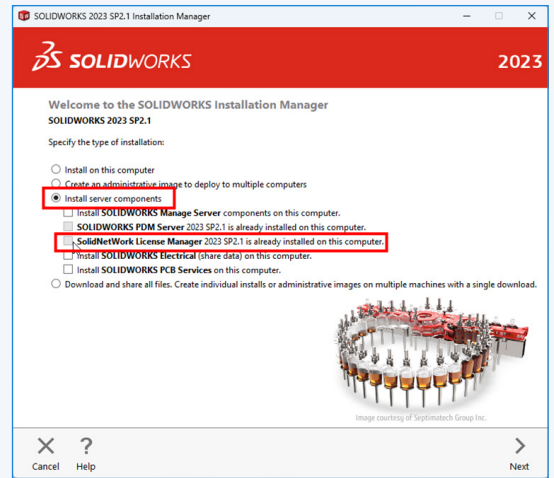
2 If you have a firewall on your server (check the one from Windows), allow the communication through port 25734 and 25735 for TCP and UDP.
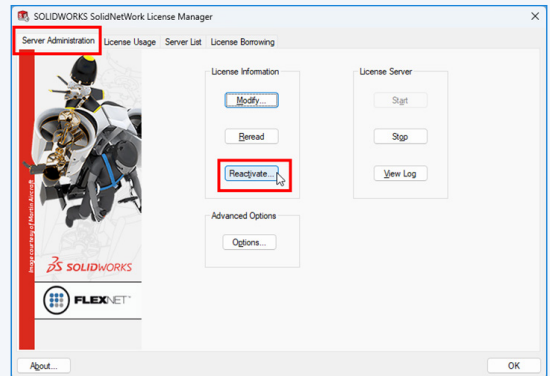
5 | Network license installation

3   Start the application SolidNetWork License Manager Server.

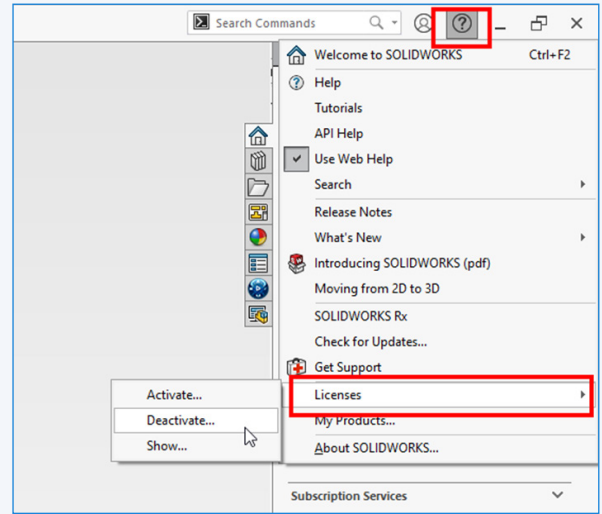**4**   You should be asked to activate your licenses: do it.

5   If not, click on the "Reactivate" button and follow the instructions.

Continue to chapter 7

6 | Standalone license installation

1 If you already have SOLIDWORKS and you want to install it on a new computer (with the same serial number), you need to disable the license on this old computer first.

2 Once the new SOLIDWORKS is installed, you will be asked to activate the license during the first start.

Continue to the next chapter
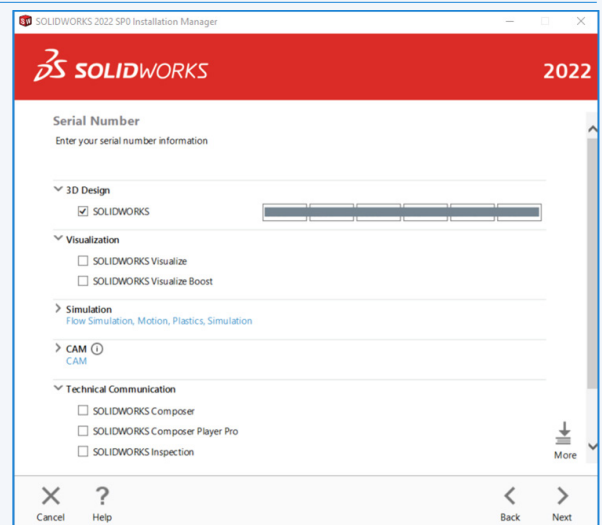
7 | Installing SOLIDWORKS

Here are some tips that will help you for a smooth SOLIDWORKS installation.
1. Ideally, the user should be admin of the computer.
2. Always start the setup.exe file with a right click and start as administrator.
3. If possible, disable the antivirus during the installation.
4. After the installation, add exceptions to your antivirus on sldworks.exe, the installation folders and on SOLIDWORKS file type (sldprt, sldasm, slddrw) to ensure a better performance.

1	In the installation files that you have just downloaded, right click on the setup.exe file and start as administrator

2	If the installation manager tells you a restart of Windows is waiting, just click OK, it's usual.

|	
3	Choose "Install on this computer".
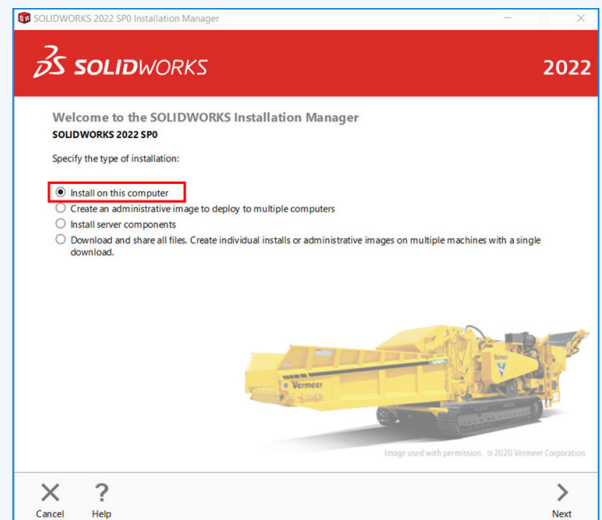
|	
4	Give your serial number: Do not enter the same number twice even if it contains several product: it is not necessary ! If you have a network license, enter it here also. For PDM, licenses are managed by the server, so nothing to enter here.

7 | Installing SOLIDWORKS

5   Choose the product you want to install:
- » If you are not sure, let everything checked.
- » Maybe uncheck SOLIDWORKS CAM if you don't do CAM.
- » Maybe uncheck Composer Player if you don't use Composer.
- » Maybe uncheck Visualize if you don't do rendering.
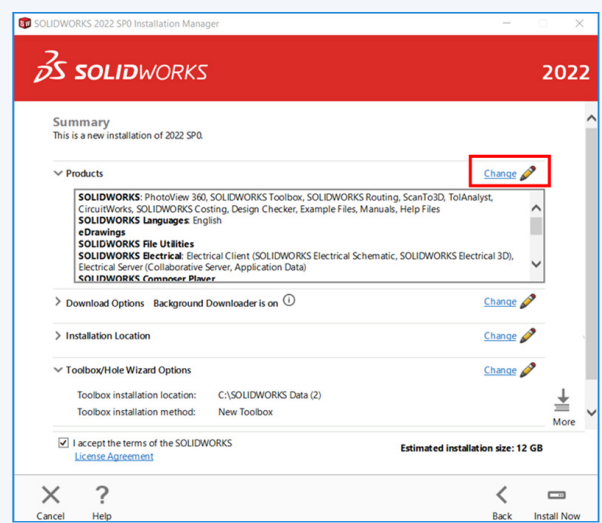
6   Installation location:
- » Let it by default except if you have several versions on the same computer.
- » If you have several versions (not advised!), add the version name in the folder name.
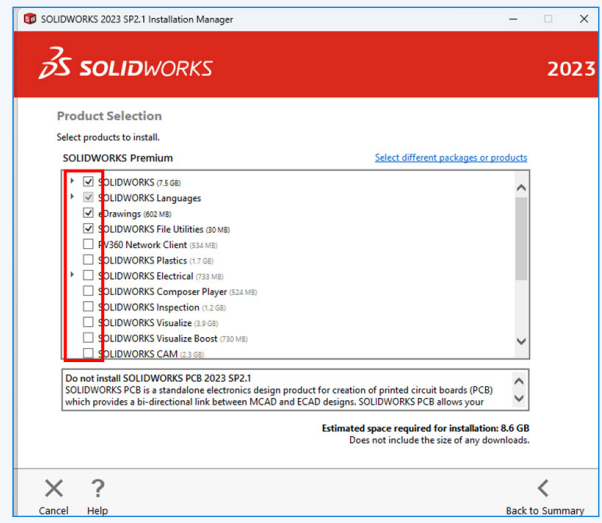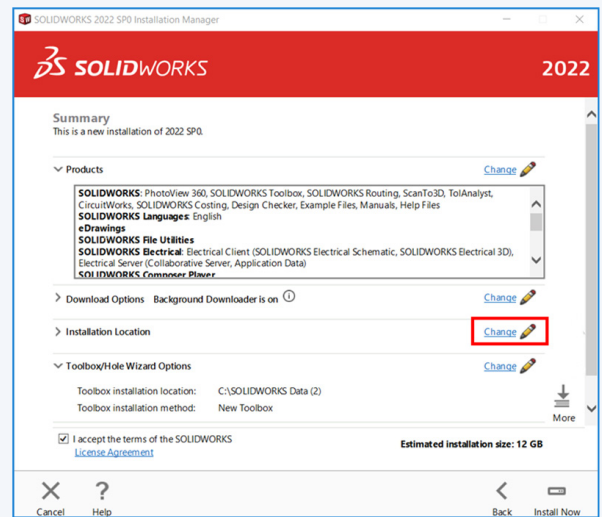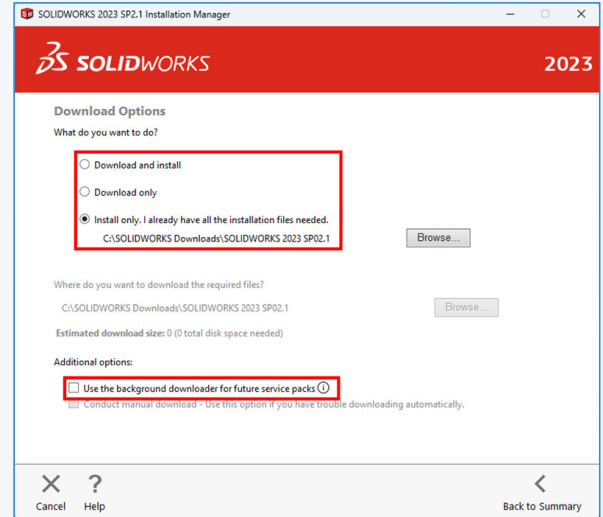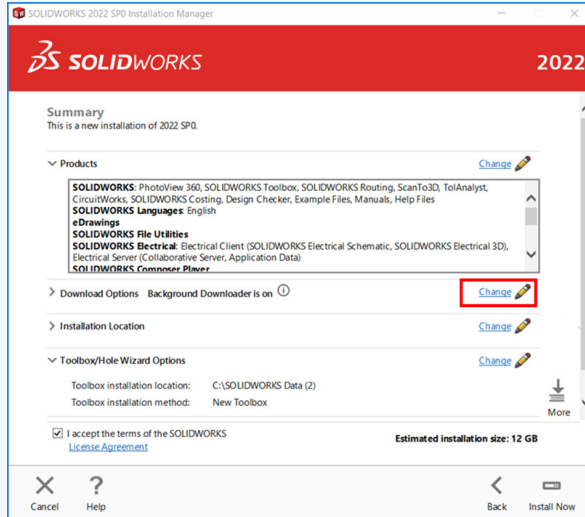
7 | Installing SOLIDWORKS

7   Toolbox
- » New installation : keep default settings (or put the name of the version into the folder name).
- » Update an existing installation:
  - » In Windows Explorer, copy your toolbox location folder and name it with the version number.
  - » In Installation Manager, choose "Use an existing SOLIDWORKS Toolbox".
  - » Point to this location in the installation manager.
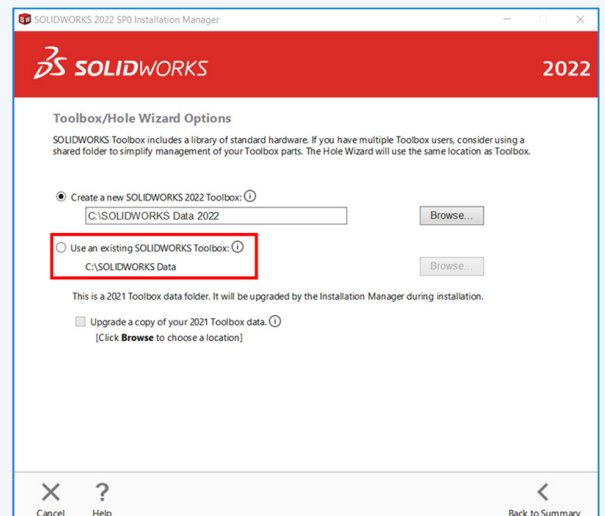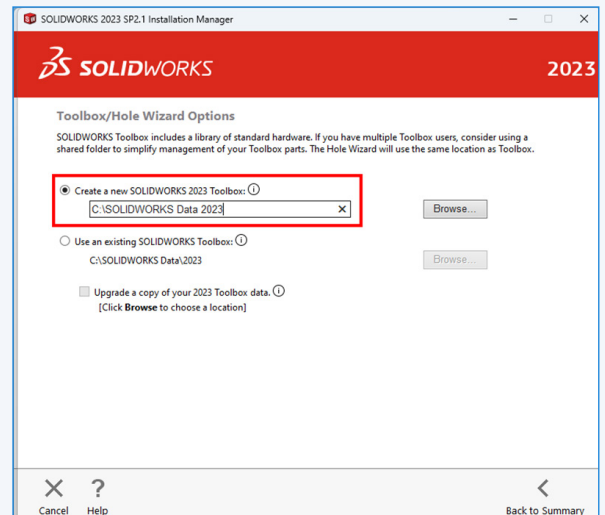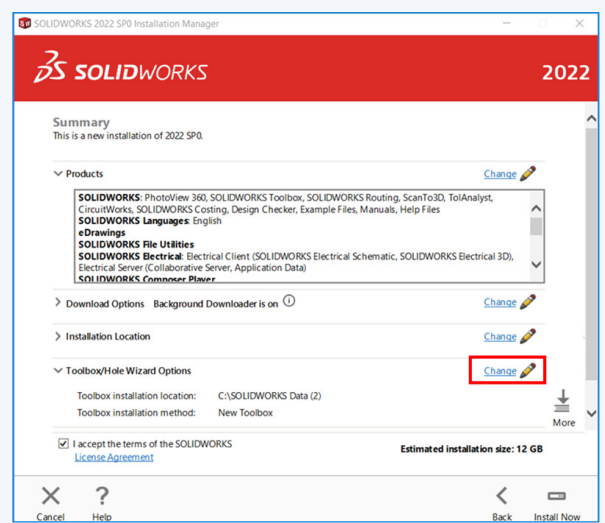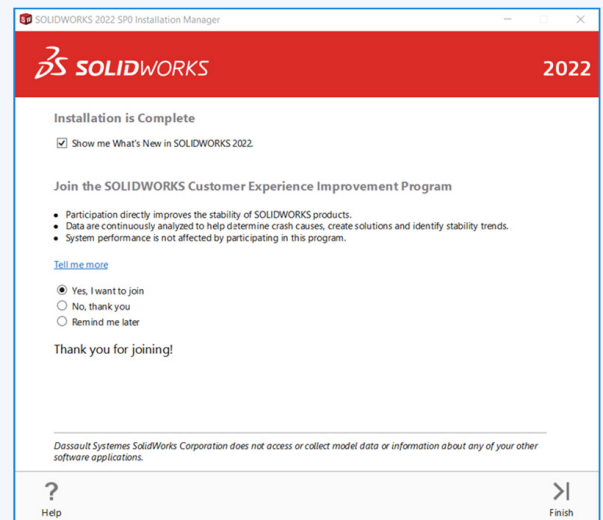
7 | Installing SOLIDWORKS

8   Background downloader: disable it.

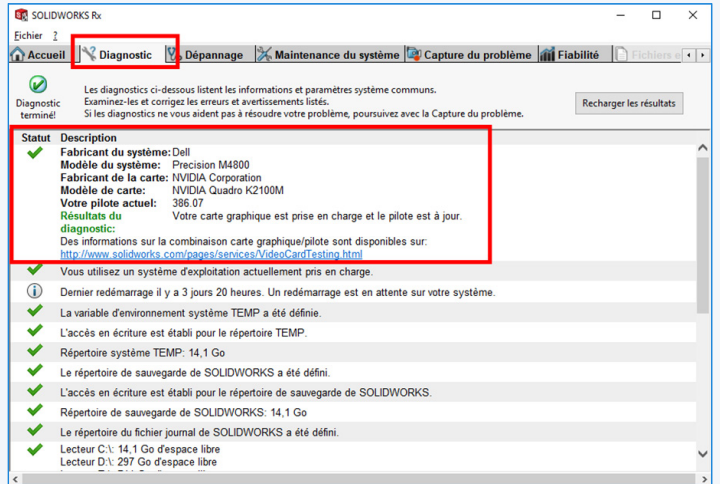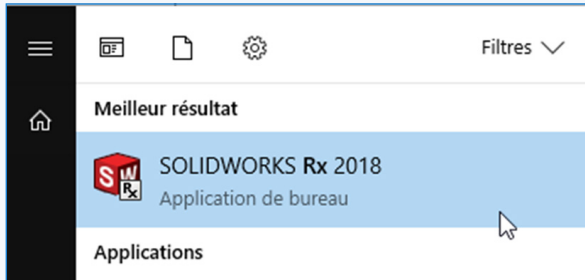9   Start the installation.

10   If you have a network license, the server address is asked:
     » 25734@ServerName (replace "ServerName" with your own server name).

11   » Installation.
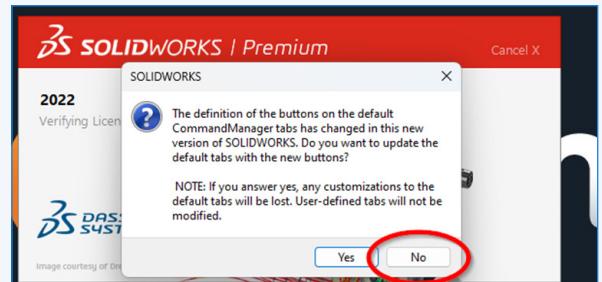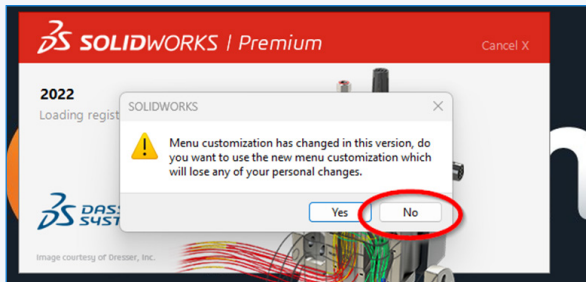
8 | After installation

1   Restart your computer.

2   Start SOLIDWORKS RX (search in Windows start menu) to check the graphic card compatibility
» Diagnostic tab.
» Update your driver if necessary.

3   Start SOLIDWORKS.

4   Active your licenses if asked (should be the case for Standalone licenses).

8 | After installation

5   After starting SOLIDWORKS, 2 questions are asked:
  - »   Say **No** to the 2 to keep ALL your customization.
 - »   Otherwise it will get back to default **new** user interface of the new version.

6   Then check in "Options", "File location", all your file location: since SOLIDWORKS keeps all previous settings, it also keeps default location pointing to previous installation folder in ProgramData\SOLIDWORKS 2022\, that we need to change to the new location since it contains the version number!
  - »   Either change manually every entry.
  - »   Or click "Edit All" and use "Find/Replace".
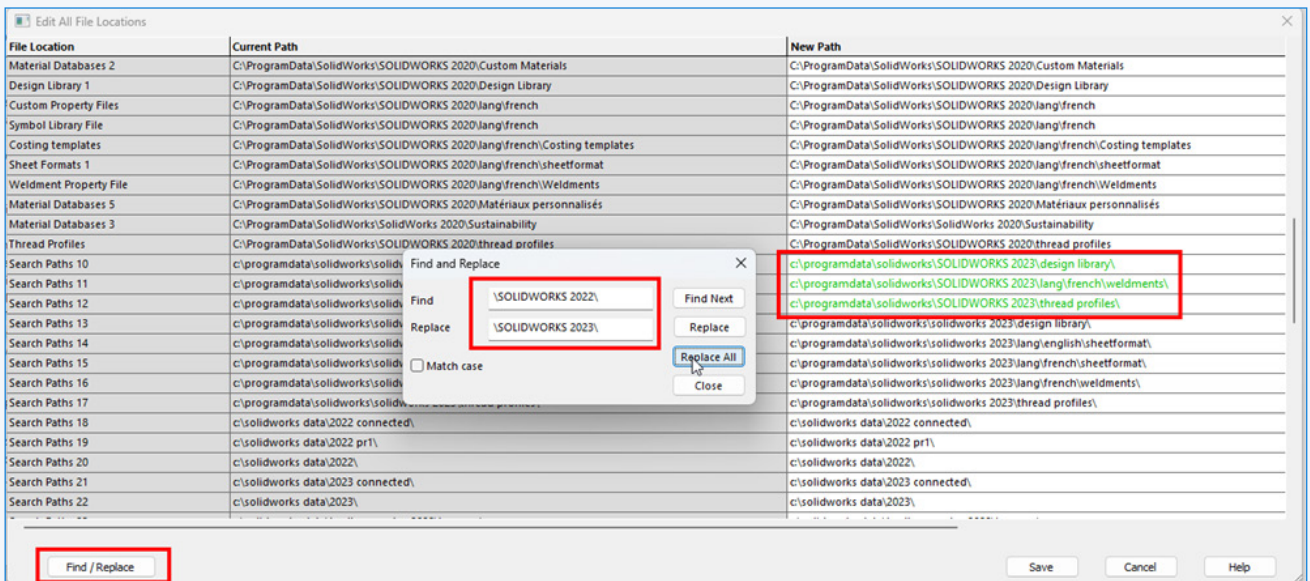
Finally, check that everything is working and save your SOLIDWORKS Settings again!
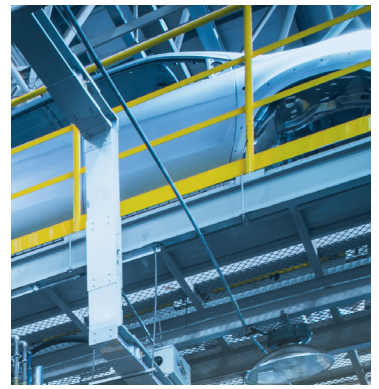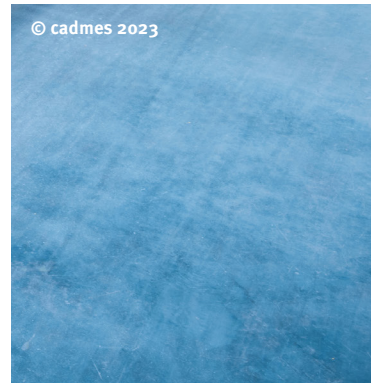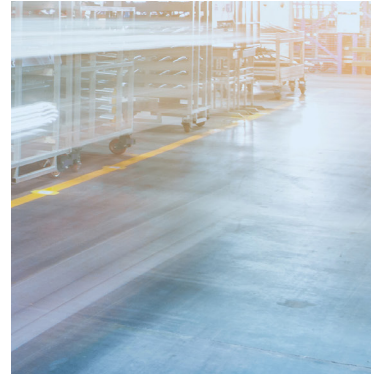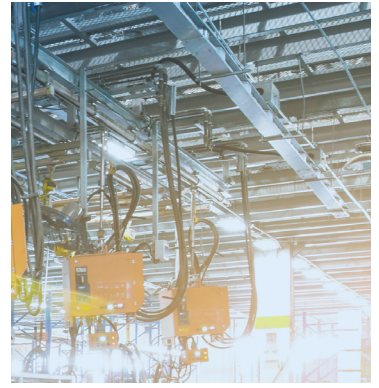
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Breaking Barriers ...

Cadmes is there for companies that want to grow, innovate, and dare to challenge the status quo.
With our expertise and experience within the industry, we can provide you with the appropriate assistance to successfully undergo transformation and work across value chains.

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