<u>Ansys Installation and troubleshooting</u>

Table of Contents

About this Document

This document provides step-by-step instructions for installing ANSYS on Linux and Windows systems within the IIT Palakkad network. This guide is intended for students, research scholars and faculty members.

Prerequisites

• The system must be connected to the IIT PKD network for the licensing.

• Administrator/root access is required.

• Minimum 50 GB free disk space and 16GB is recommended.

• Required OS libraries and packages must be installed

For windows Operating System, please refer to the link given below regarding the prerequisites

https://ansyshelp.ansys.com/public/account/secured?returnurl=/Views/Secured/corp/v251/en/installation/win_swprereqs.html

For Linux based Operating Systems, please refer to the link given below

https://ansyshelp.ansys.com/public/account/secured?returnurl=/Views/Secured/corp/v251/en/installation/unix_prereq.html

The specific linux libraries needed for the Ansys installer to run is given below https://ansyshelp.ansys.com/public/account/secured?returnurl=/Views/Secured/corp/v252/en/installation/unix_platform_libraries.html

Download Instructions

ANSYS installation packages are hosted on an internal institute server and are accessible only from within the IIT Palakkad network.

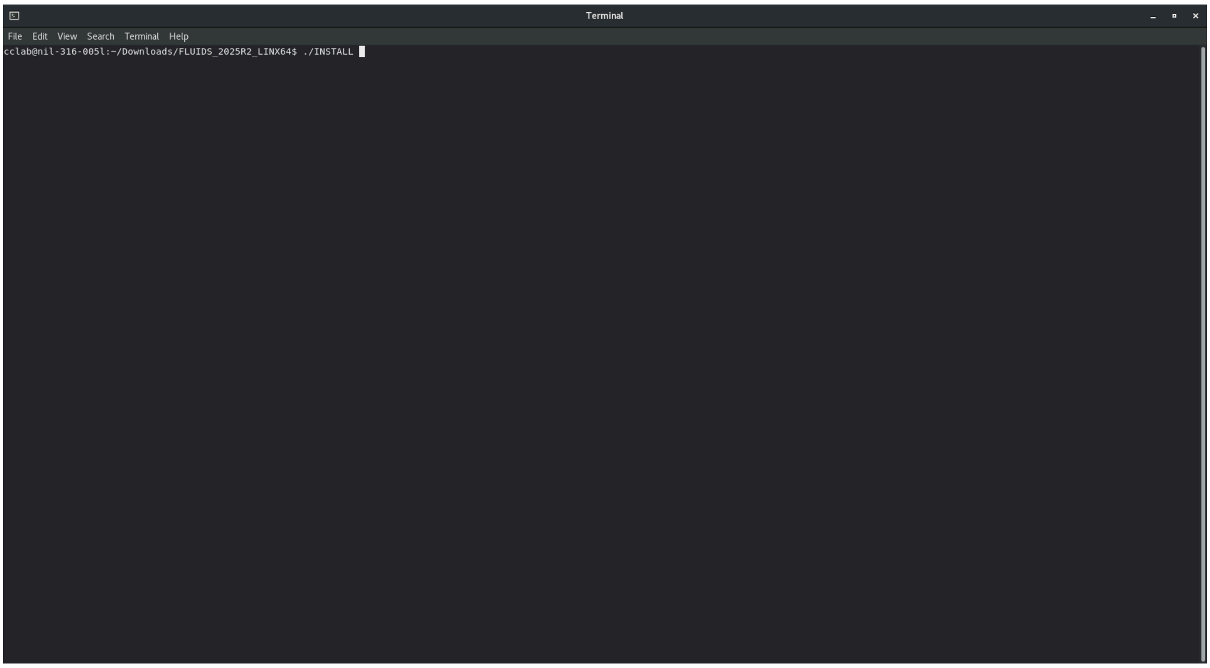
You can download the setup for Ansys respective version for your OS from the following link

http://10.128.7.230/index.php/s/AddtKBRJmXLJr8a

Installation of Ansys 2025R2 Fluids (Linux)

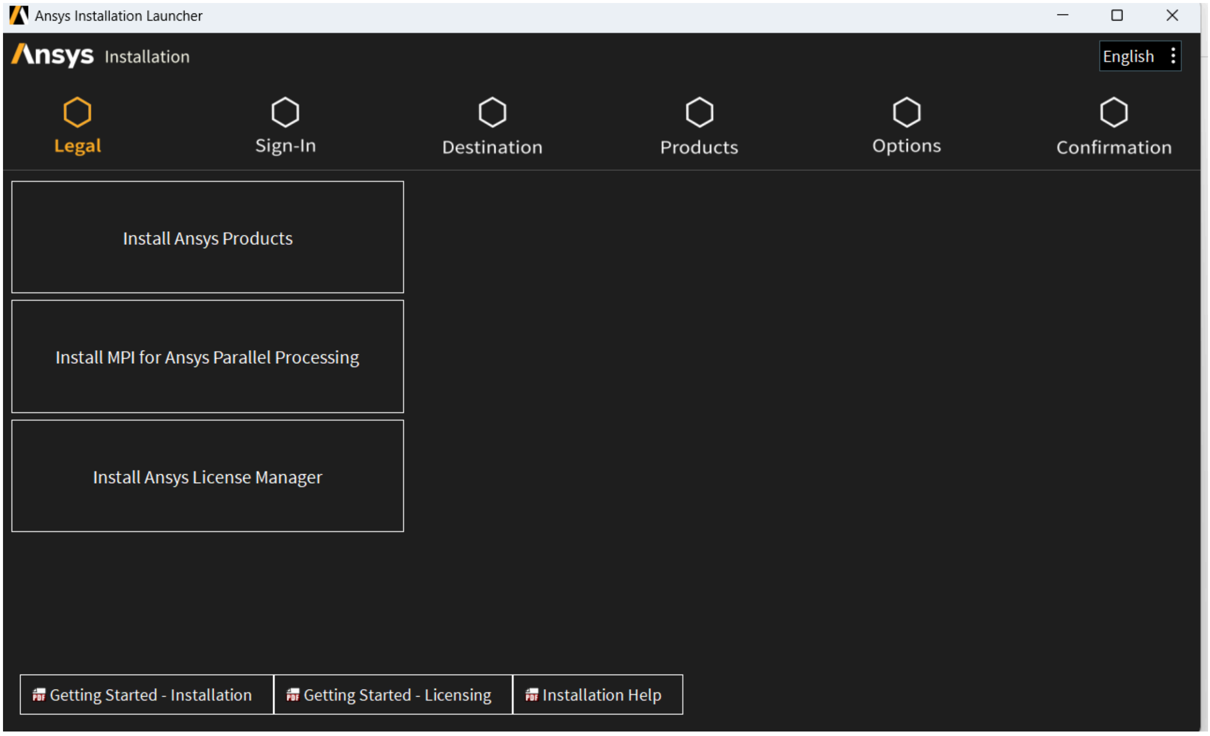
Step 1: Download the setup file and extract it. Open the extracted folder using Terminal and execute the following command

./INSTALL

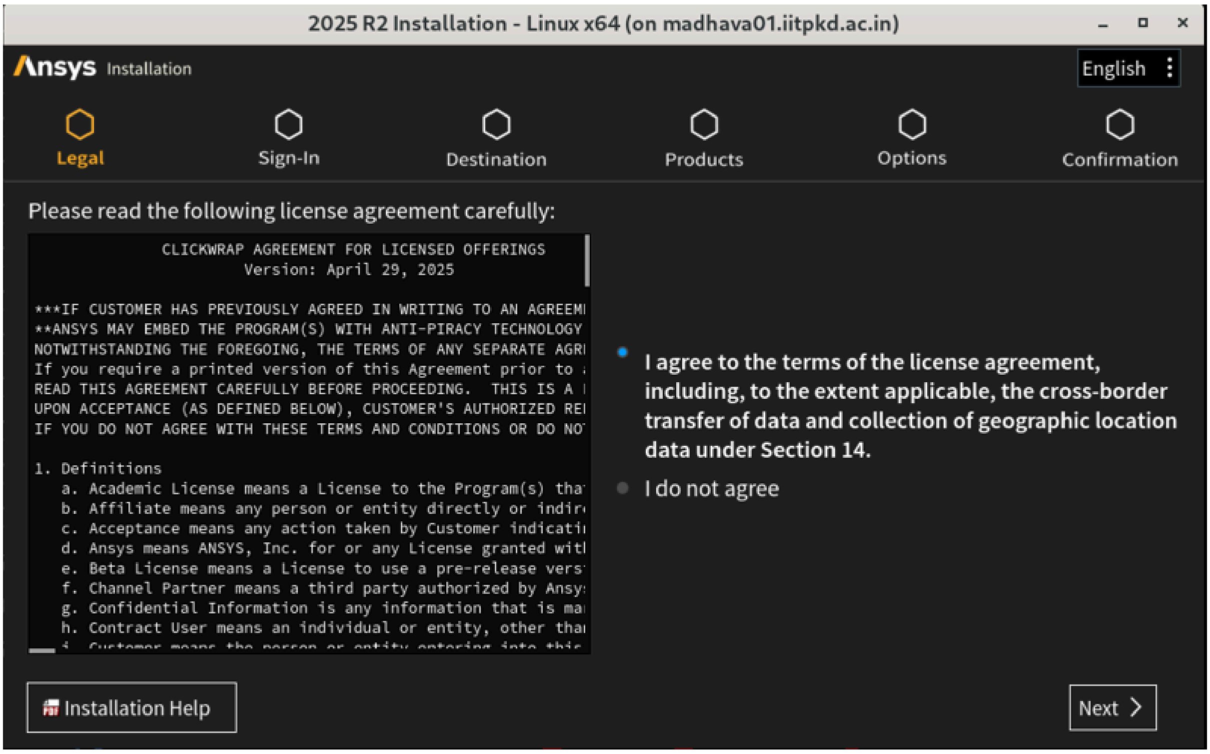


If all the dependent packages were installed previously, then the Ansys installer will open as shown in Step 2 and beyond

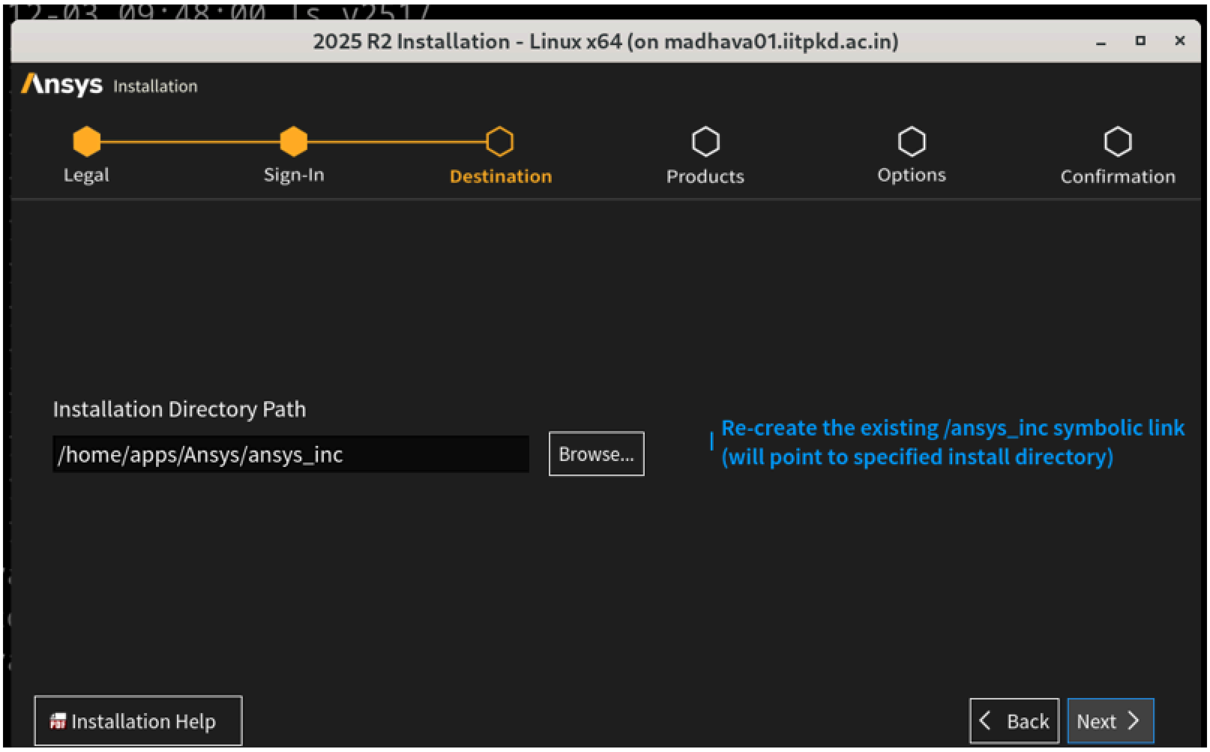
Step 2: Click on install ansys products



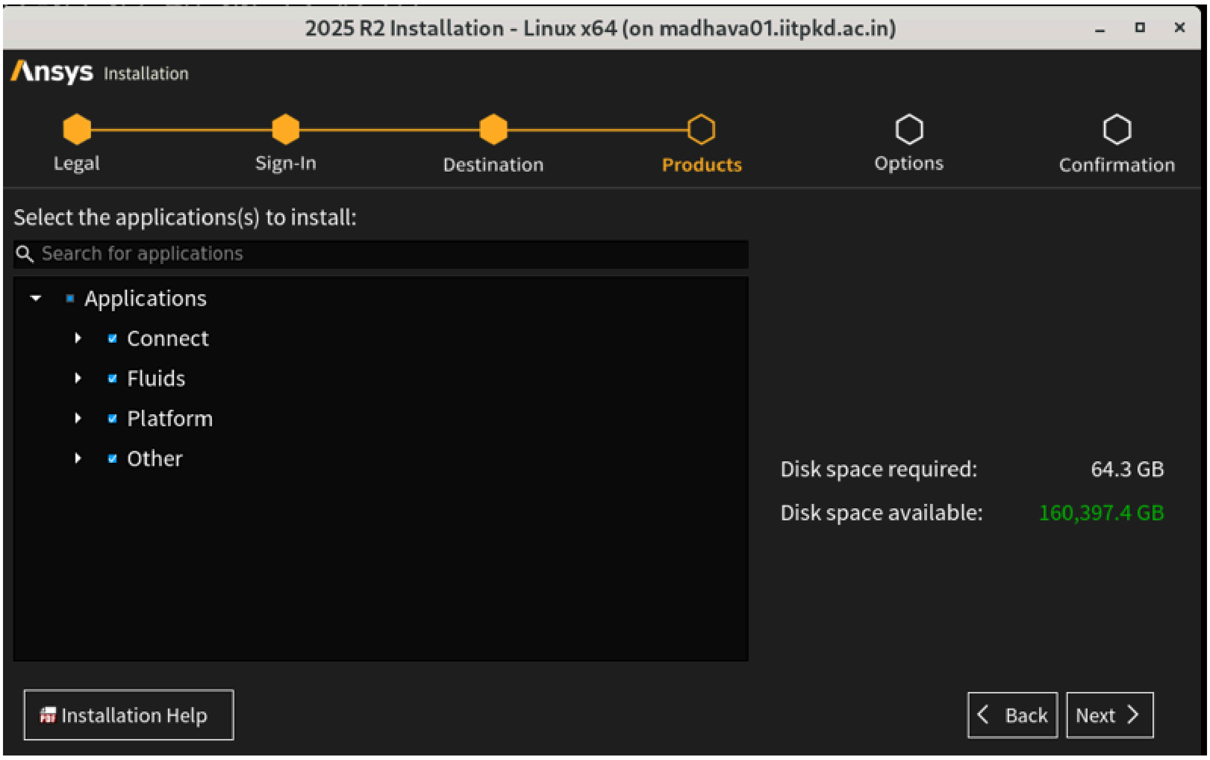
Step 3: Click on I agree



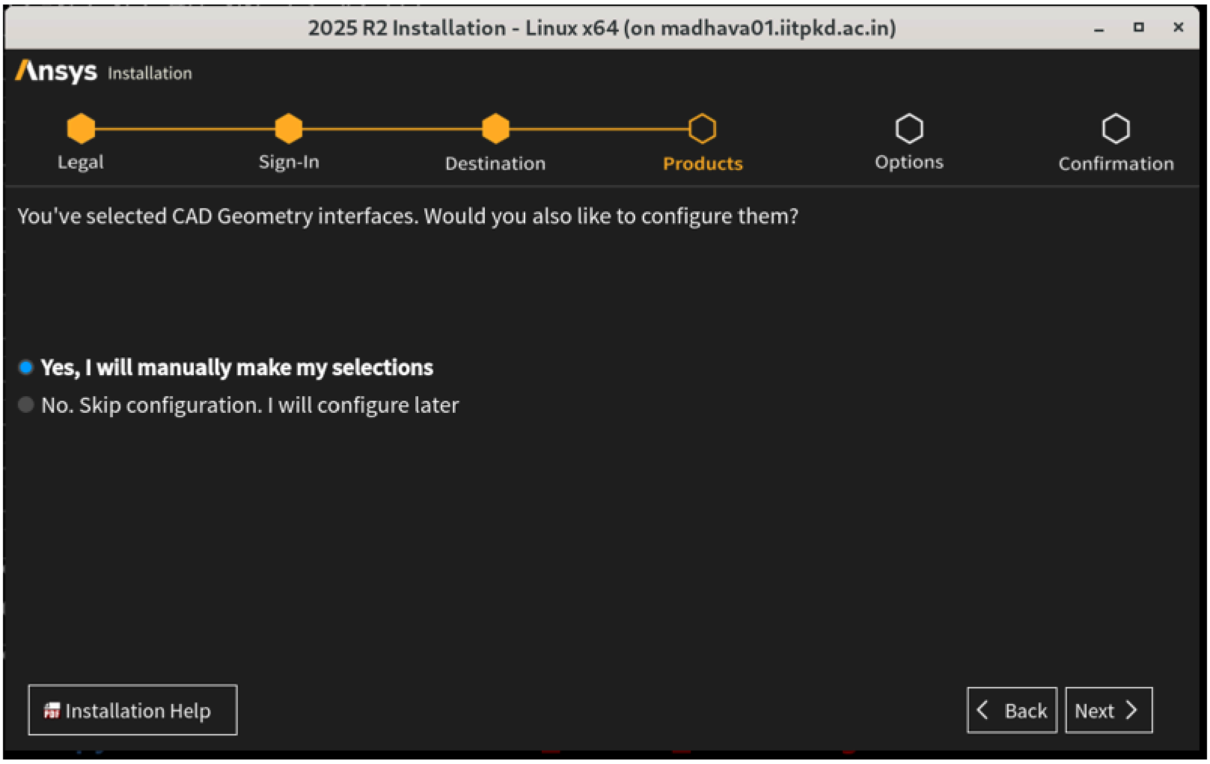
Step 4: You can choose the default path or install it to a different path if needed



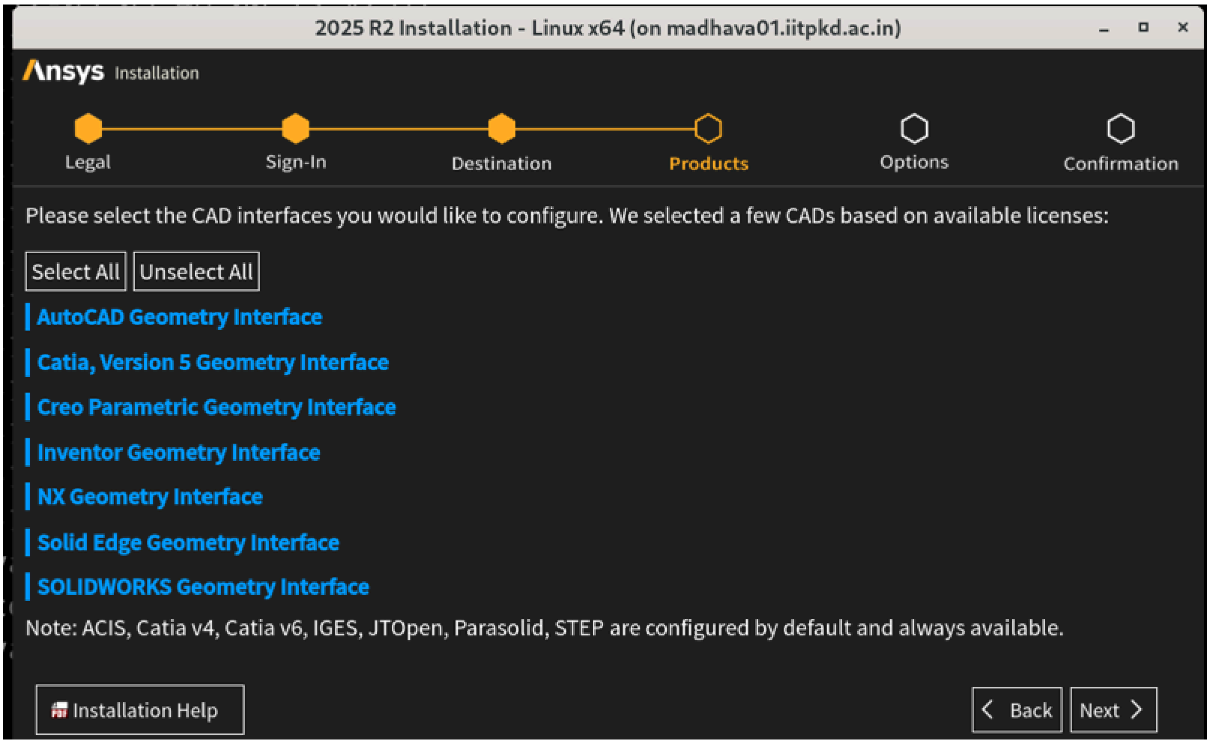
Step 5: Select the desired applications



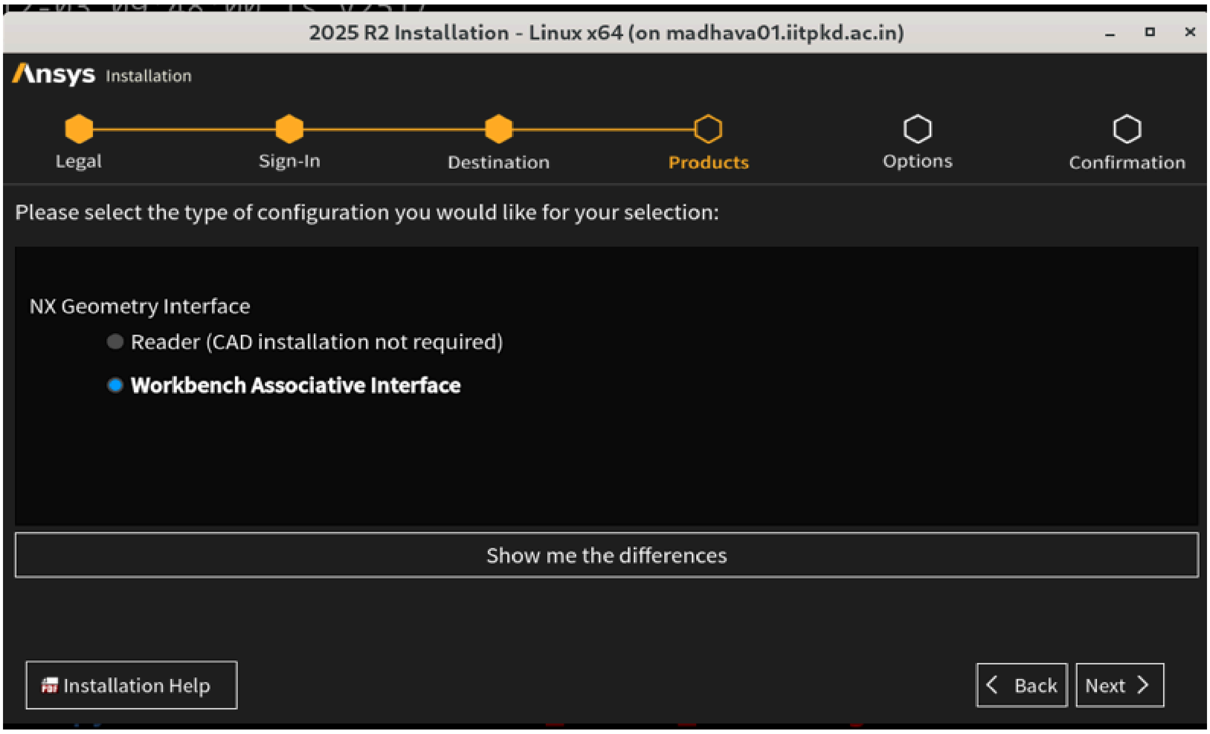
Step 6: Select Yes, I will manually configure interfaces and Click Next



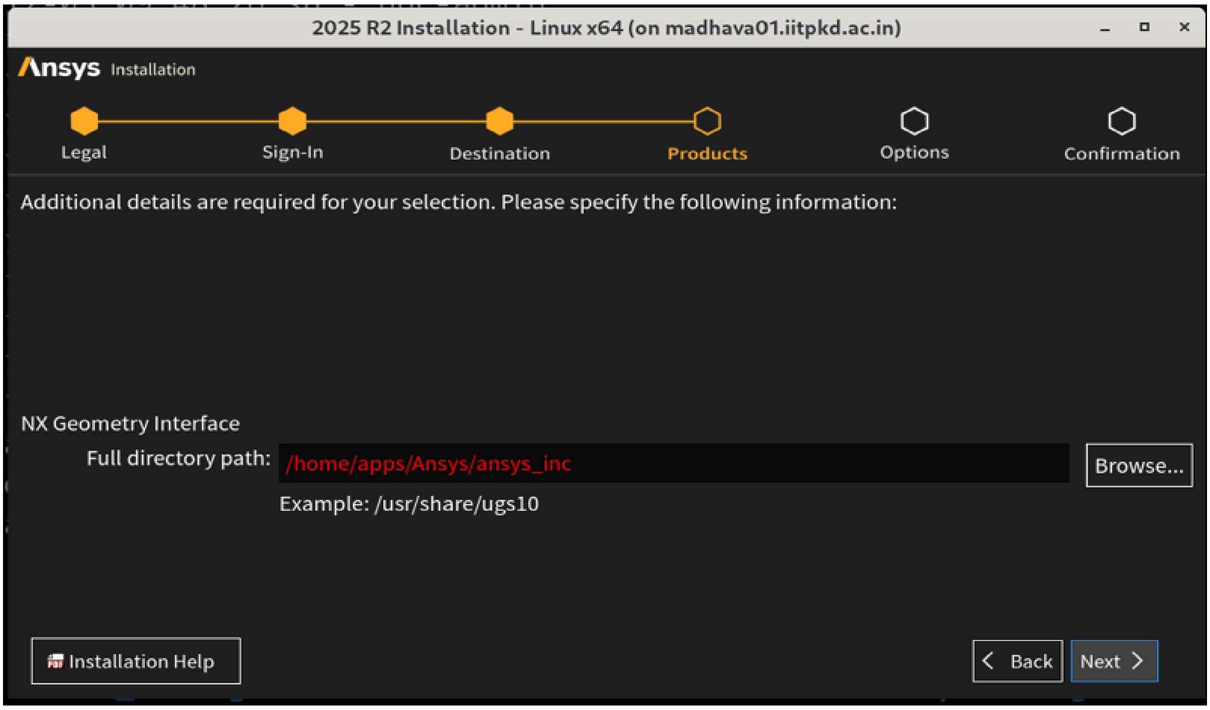
Step 7: Choose the CAD interfaces required



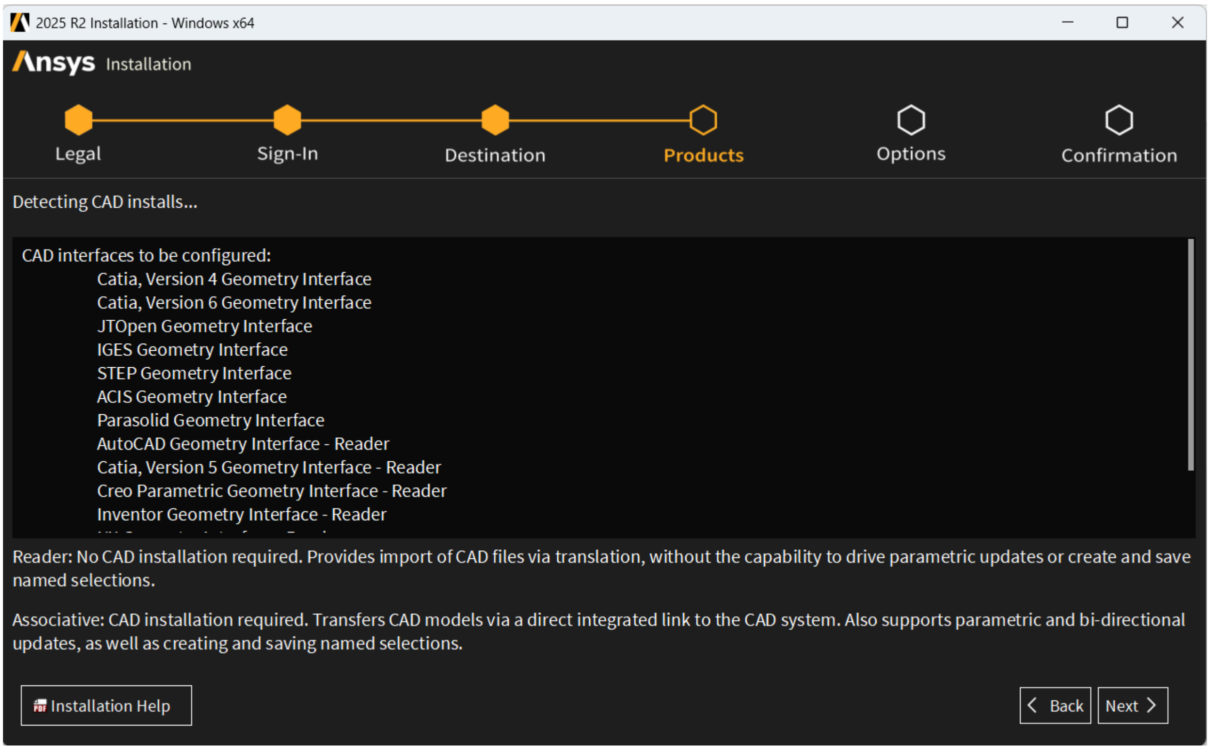
Step 8: If you have selected NX Geometry Interface in the previous step, then select the type of configuration for NX Interface and Click Next. If it is not selected , then click next and move to the installation (Step 11)



Step 9: Set the path for installing NX Geometry Interface. It is the path where ansys is installed(Path is given in Step 4)

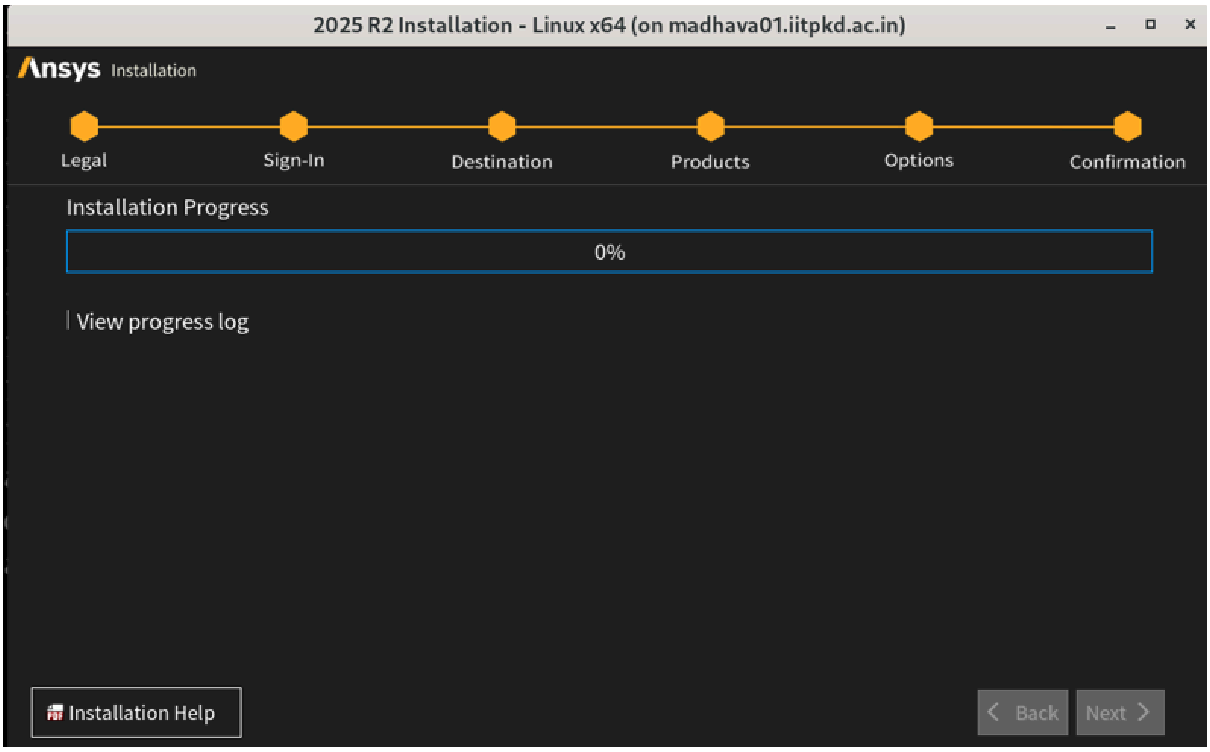


Step 10:Click Next

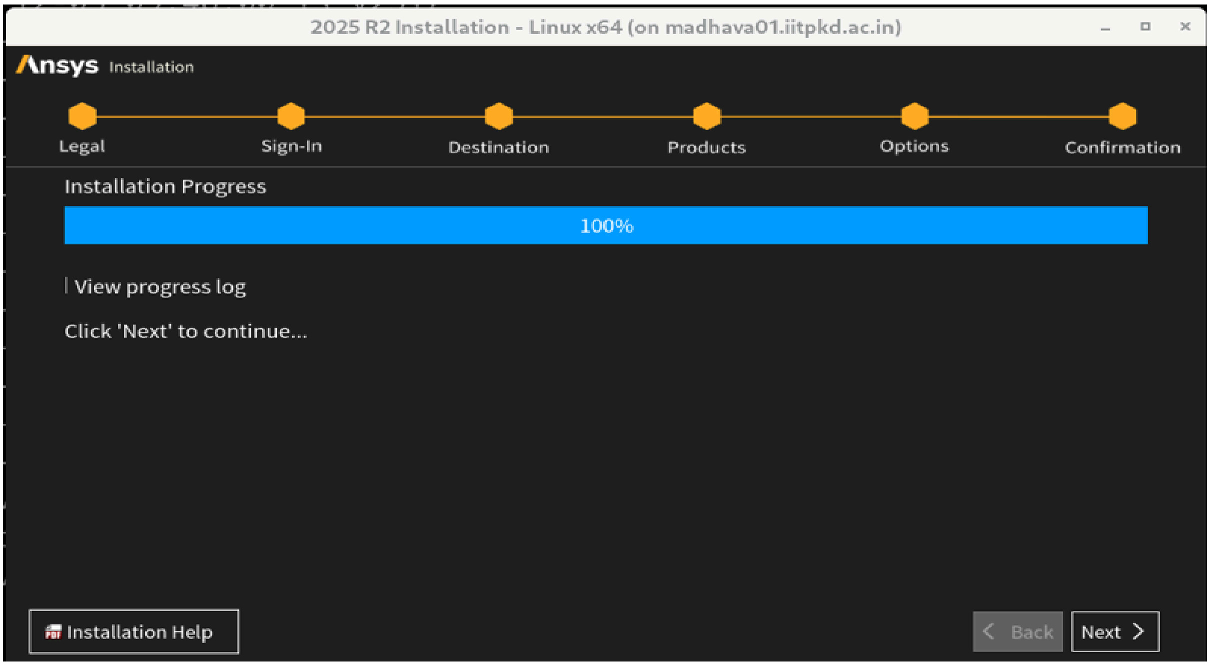


Step 11: Click next

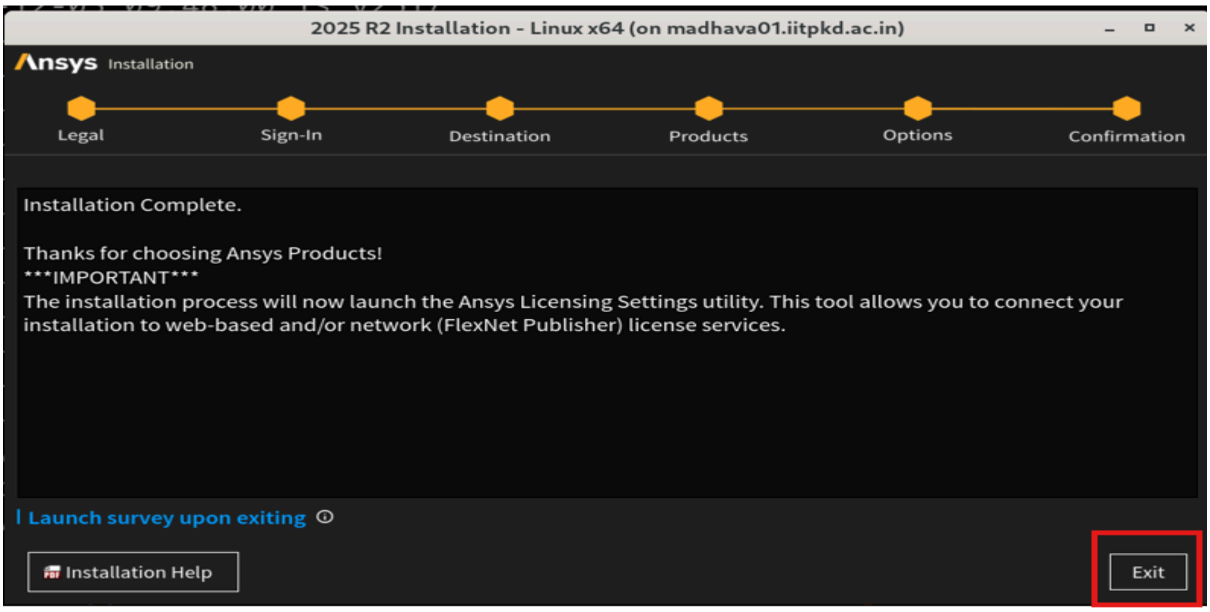
Now Installation has started. Wait for the installation to complete



Step 12: Installation is completed. Click on “Next”



Step 13: Click on Exit



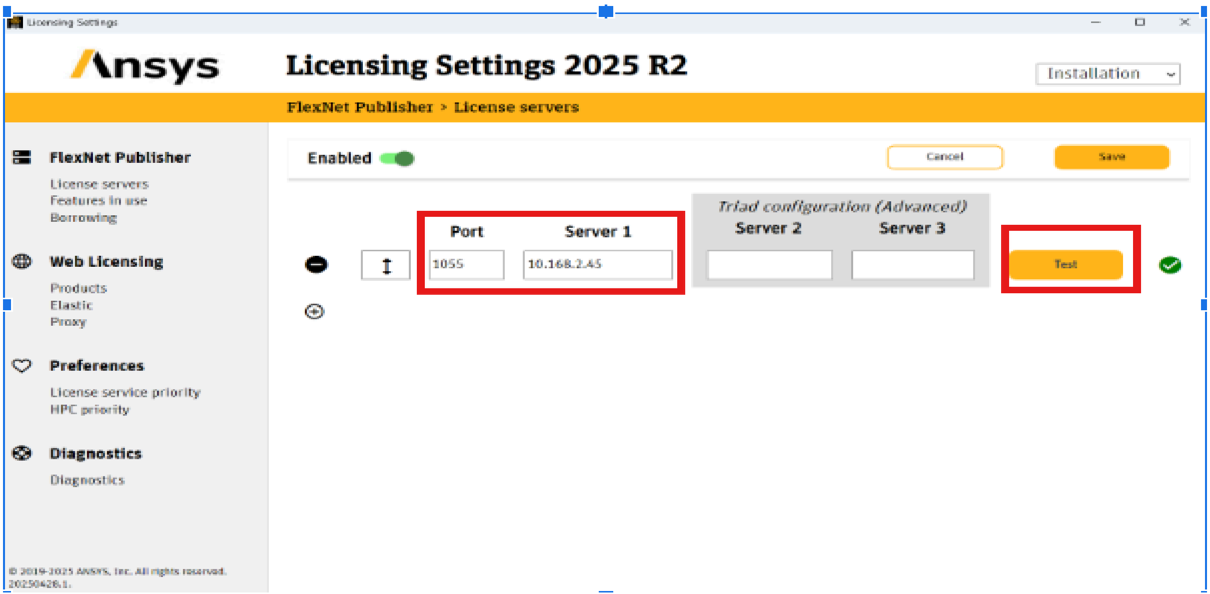
Step 14: License Manager WindowOpens up.

Assign the Port and IP as the following

Port: 1055

IP:10.168.2.45

Click on the Test button. If the connection is successful, a green tick mark appears and make sure to Click Save button.



This completes the installation process.

Installation of Ansys 2025R2 Structures(Windows)

If you are already using Ansys 18.2, uninstall it before proceeding with the installation of Ansys 2025. Ansys 18.2 can be uninstalled by going to the following path

C:\ProgramData\Microsoft\Windows\Start Menu\Programs\ANSYS 18.2

Click on the uninstall icon and proceed with the uninstallation

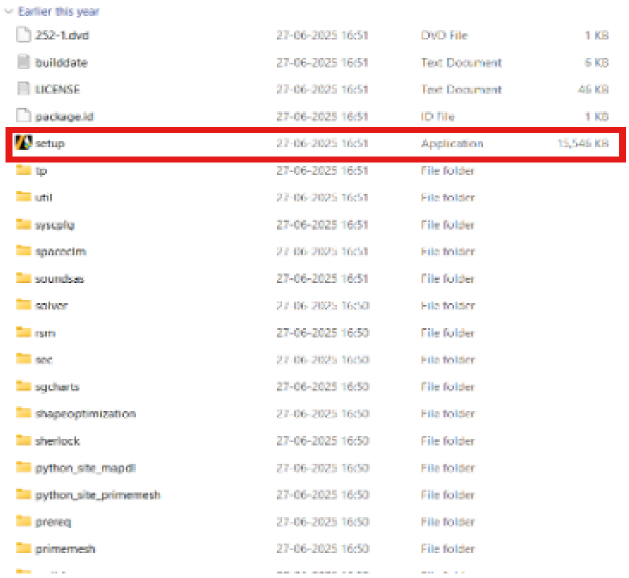
You can **download** the setup for Ansys 2025R2 from the following link

http://10.128.7.230/index.php/s/AddtKBRJmXLJr8a

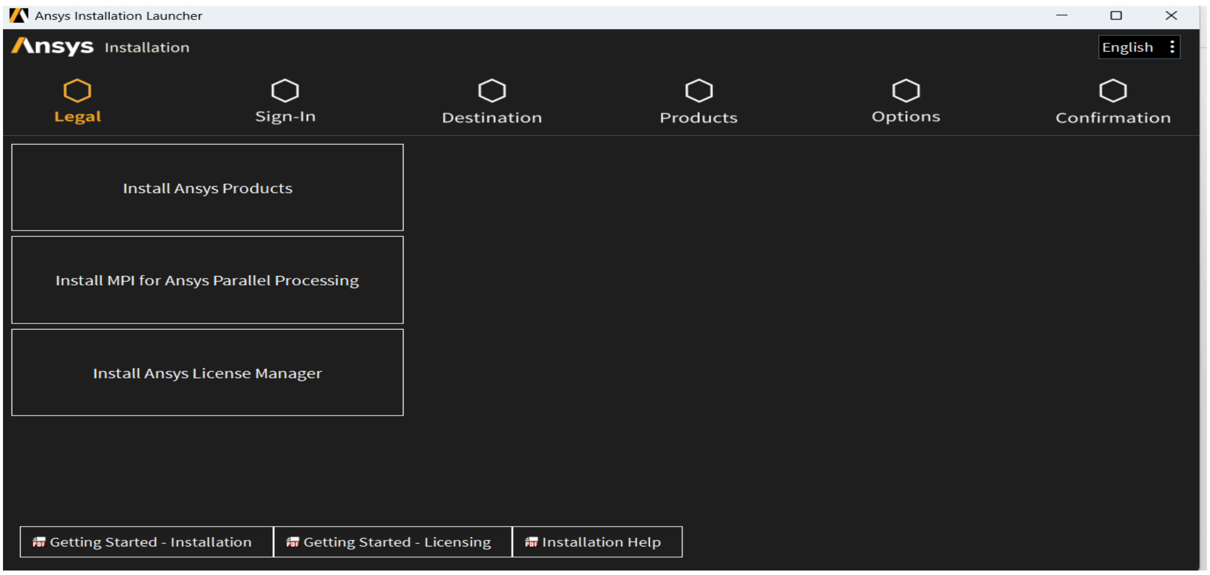
In the above link, go to the folder Windows_2025R2 and download STRUCTURES_2025R2_WINX64.zip

Note: The above link is internal, can only be accessed using institute network

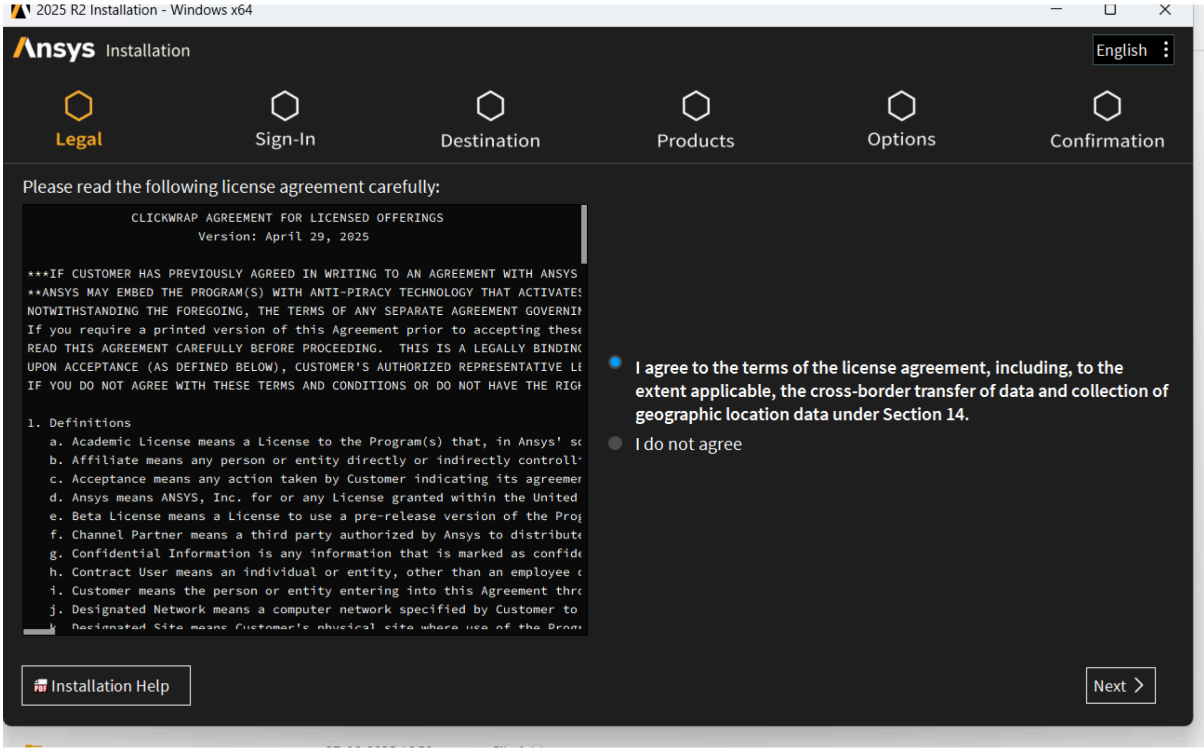
Step 1: Download the setup file and extract it. In the extracted folder double click the setup option as shown in the screenshot below:



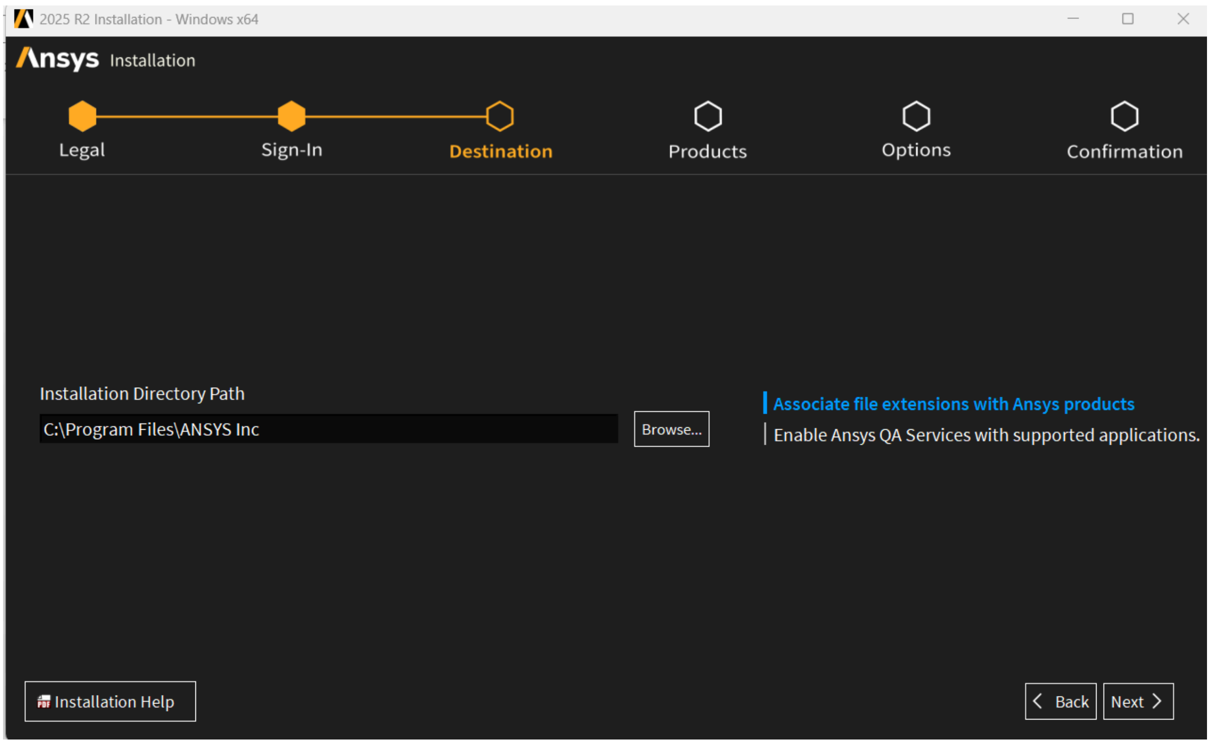
Step 2: Click on install ansys products



Step 3: Click on I agree

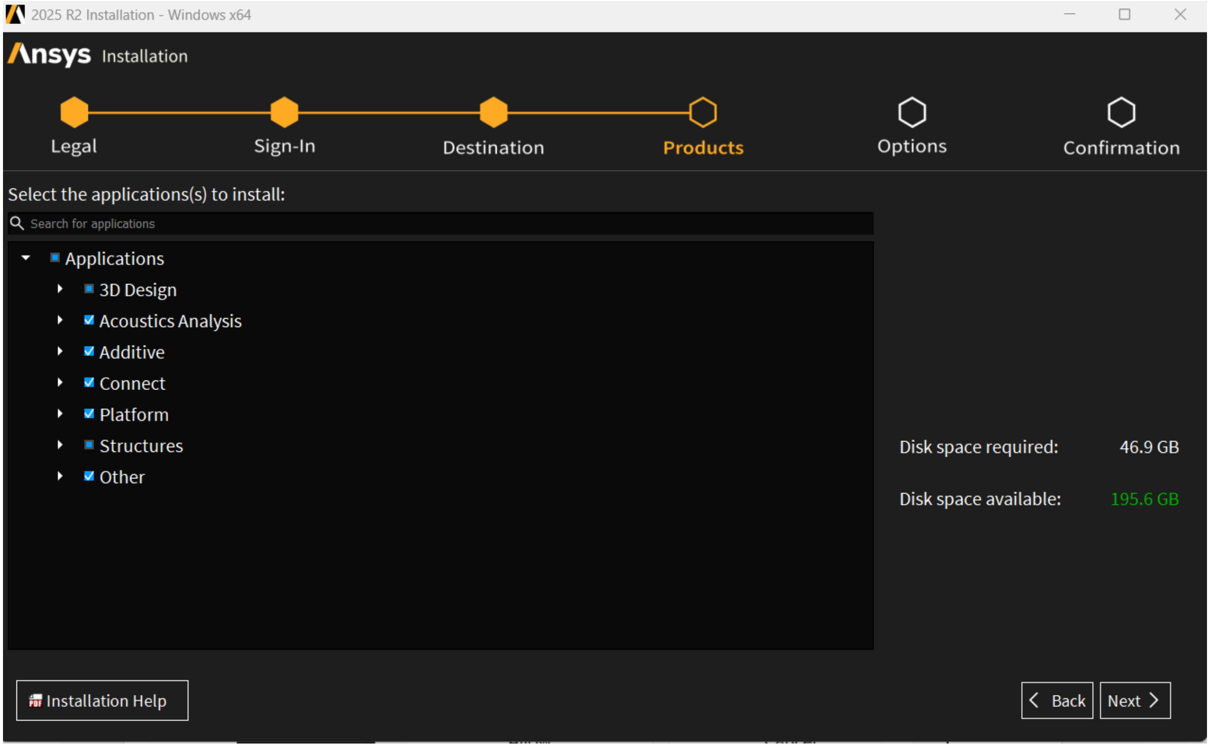


Step 4: You can choose the default path or install it to a different path if needed

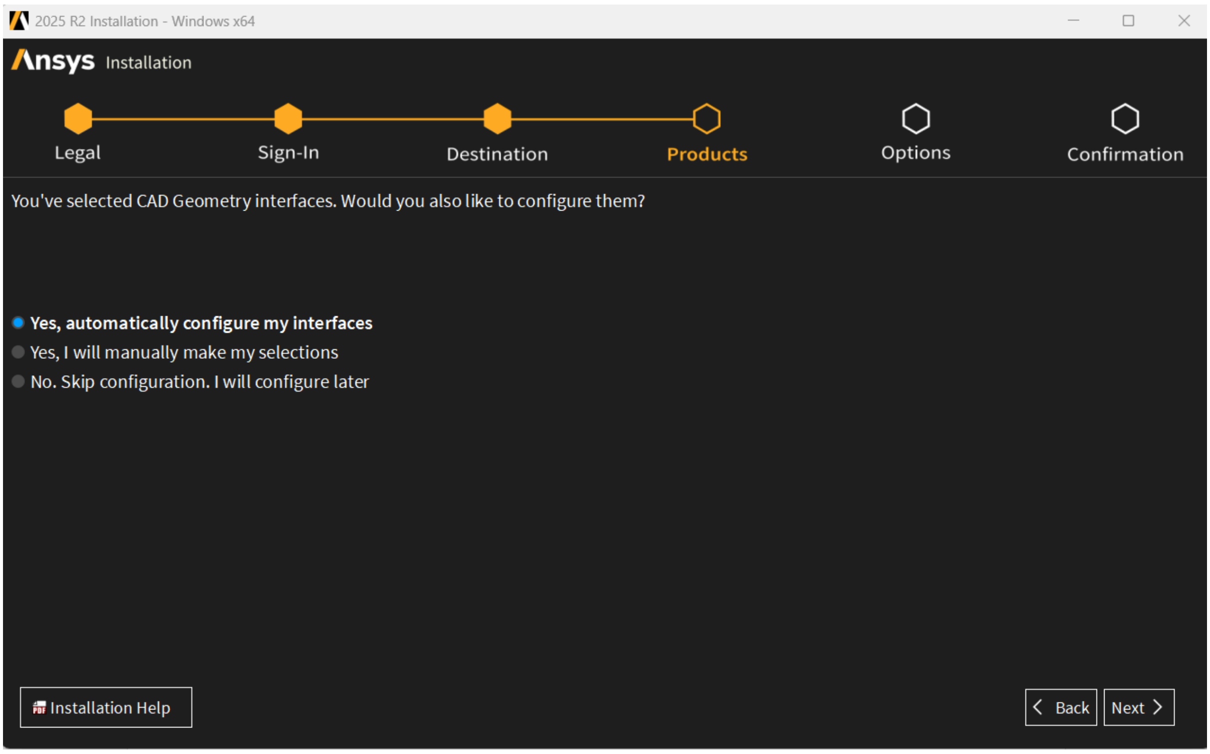


Step 5: Click on Allow to give firewall permission

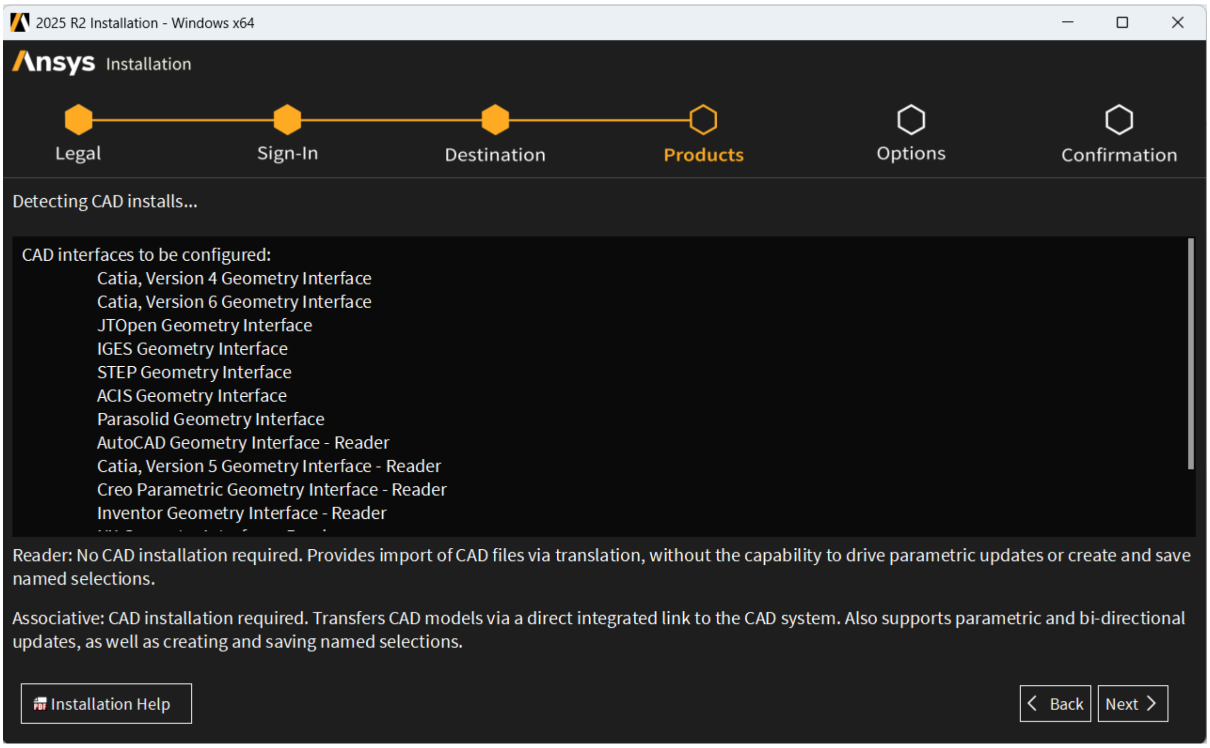
Step 6: Select the desired applications



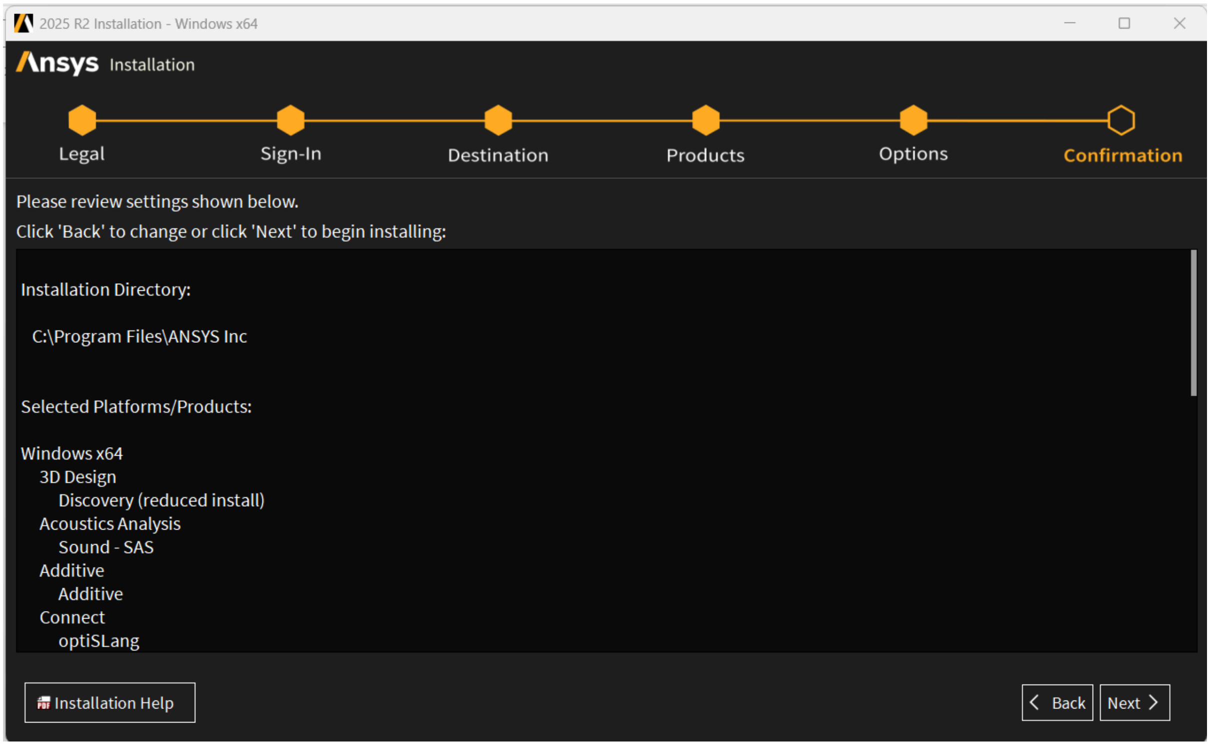
Step 7: Select automatically configure interfaces and Click Next



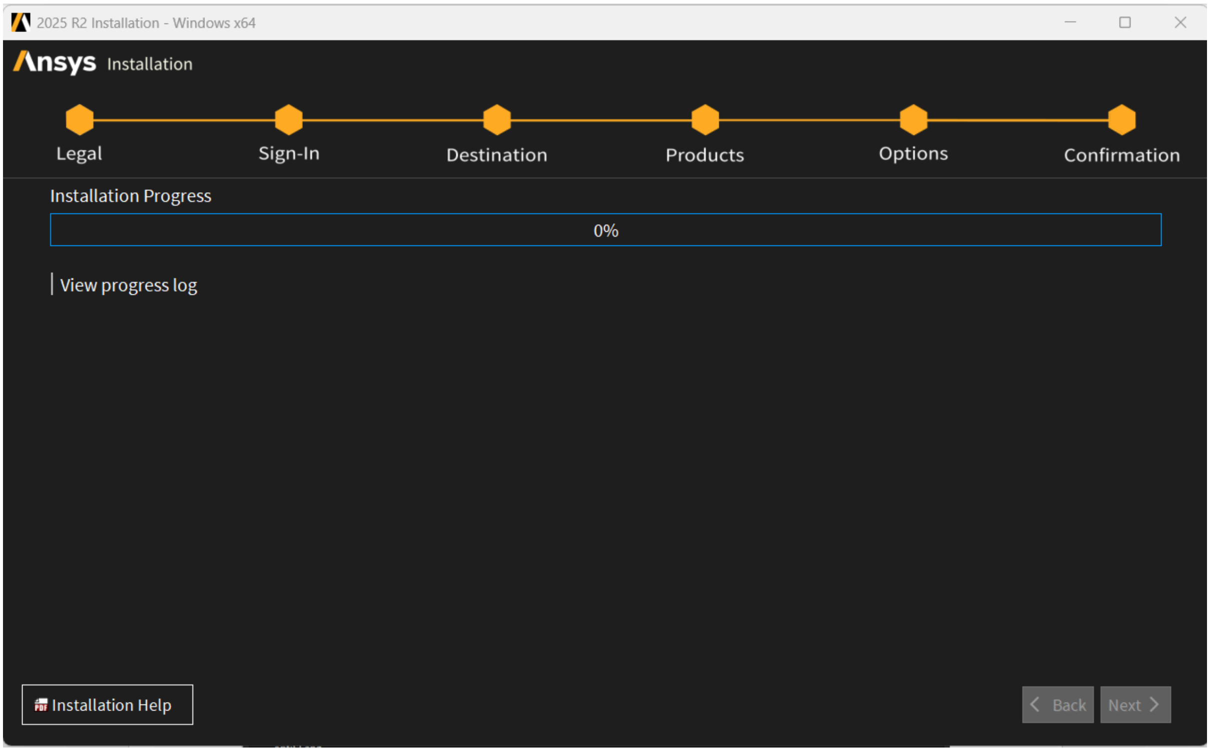
Step 8: Click Next



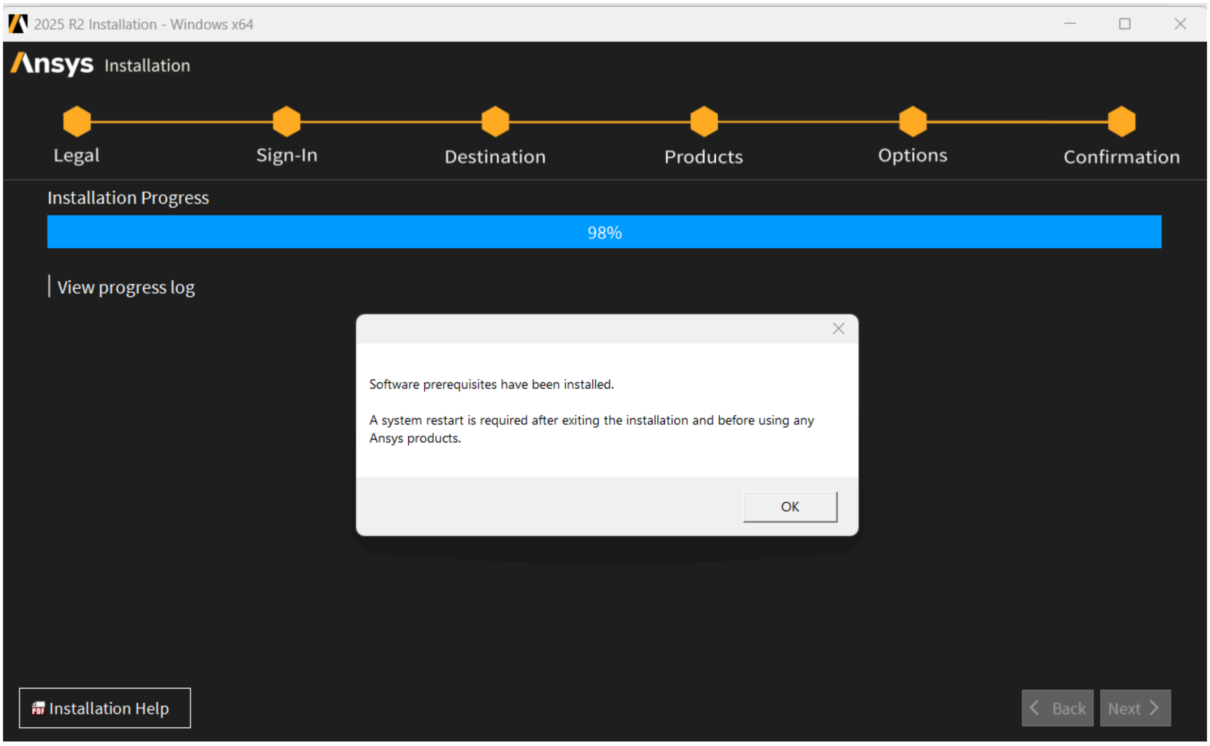
Step 9: Click next



Now Installation has started. Wait for the installation to complete



Step 10: Installation is completed. Click on “OK” in the dialogue box



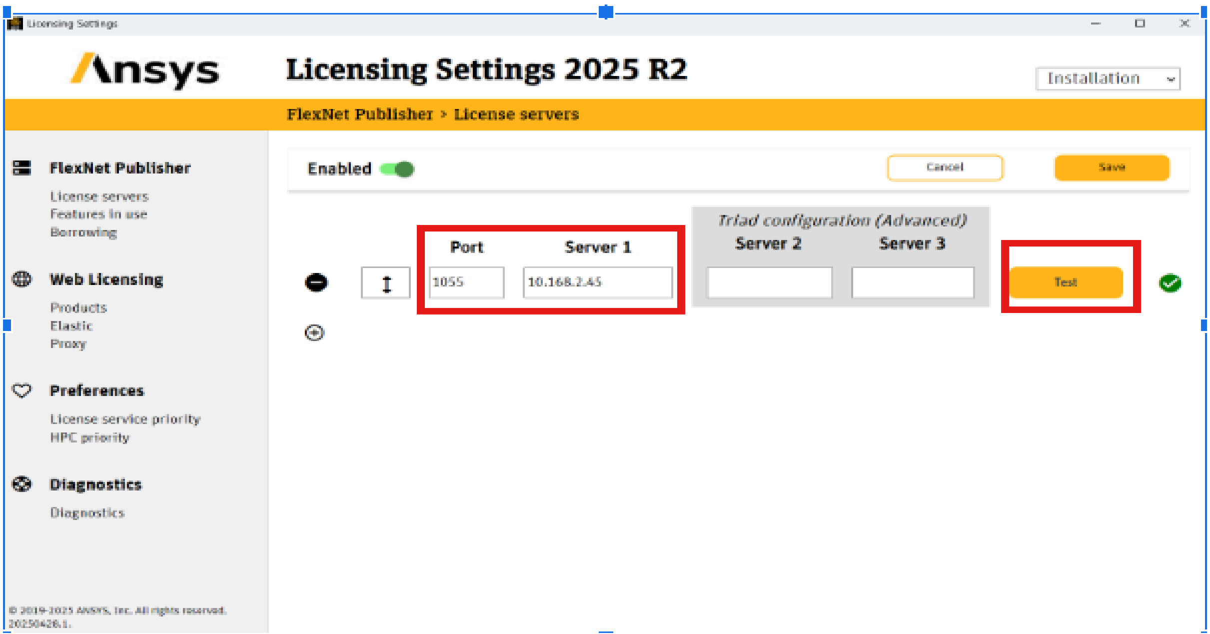
Step 11: License Manager WindowOpens up.

Assign the Port and IP as the following

Port: 1055

IP:10.168.2.45

Click on the Test button. If the connection is successful, a green tick mark appears. Do not forget to click Save button



This completes the installation process. You can the open Workbench from the startup menu

Error condition: If you are getting the error message "Could not set flexNet servers" while clicking on the Save button, then go to

C:\Program Files\ANSYS Inc\Shared Files\Licensing\ansyslmd.ini

In this file set the content as given below:

SERVER=1055@10.168.2.45

Installation of Ansys 2025R2 NCode(Windows)

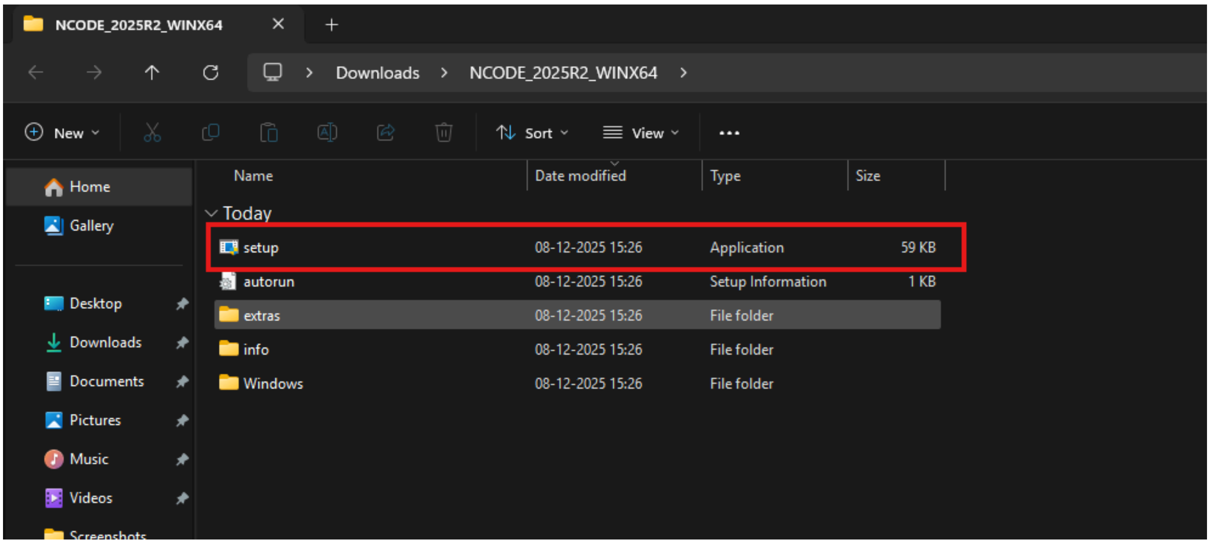
You can download the setup for Ansys 2025R2 from the following link

http://10.128.7.230/index.php/s/AddtKBRJmXLJr8a

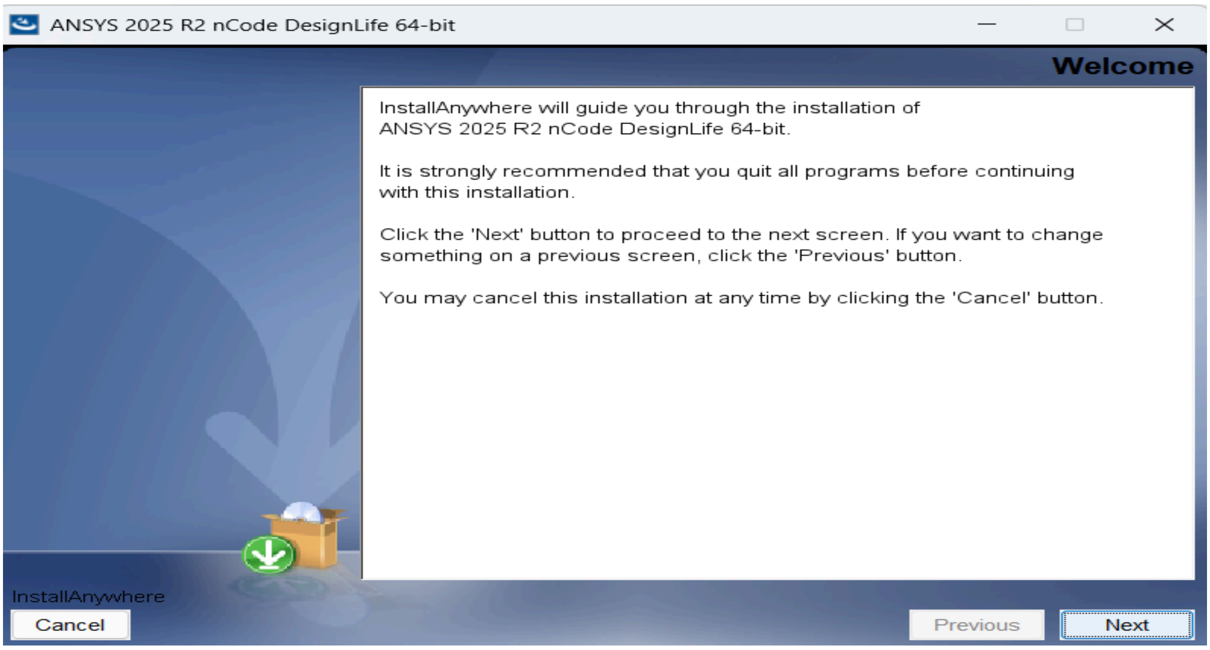
In the above link, go to the folder Windows_2025R2 and download NCODE_2025R2_WINX64.zip

Note: The above link is internal, can only be accessed using institute network

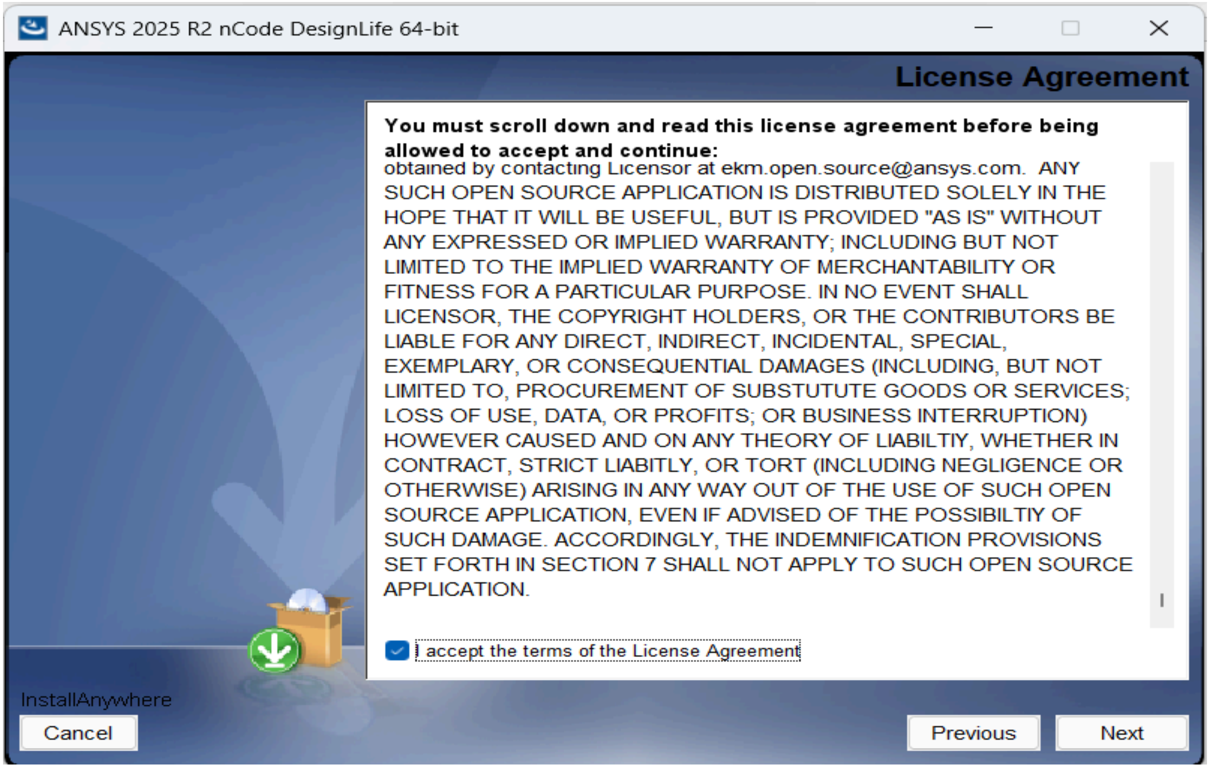
Step 1: After extracting, double click on setup



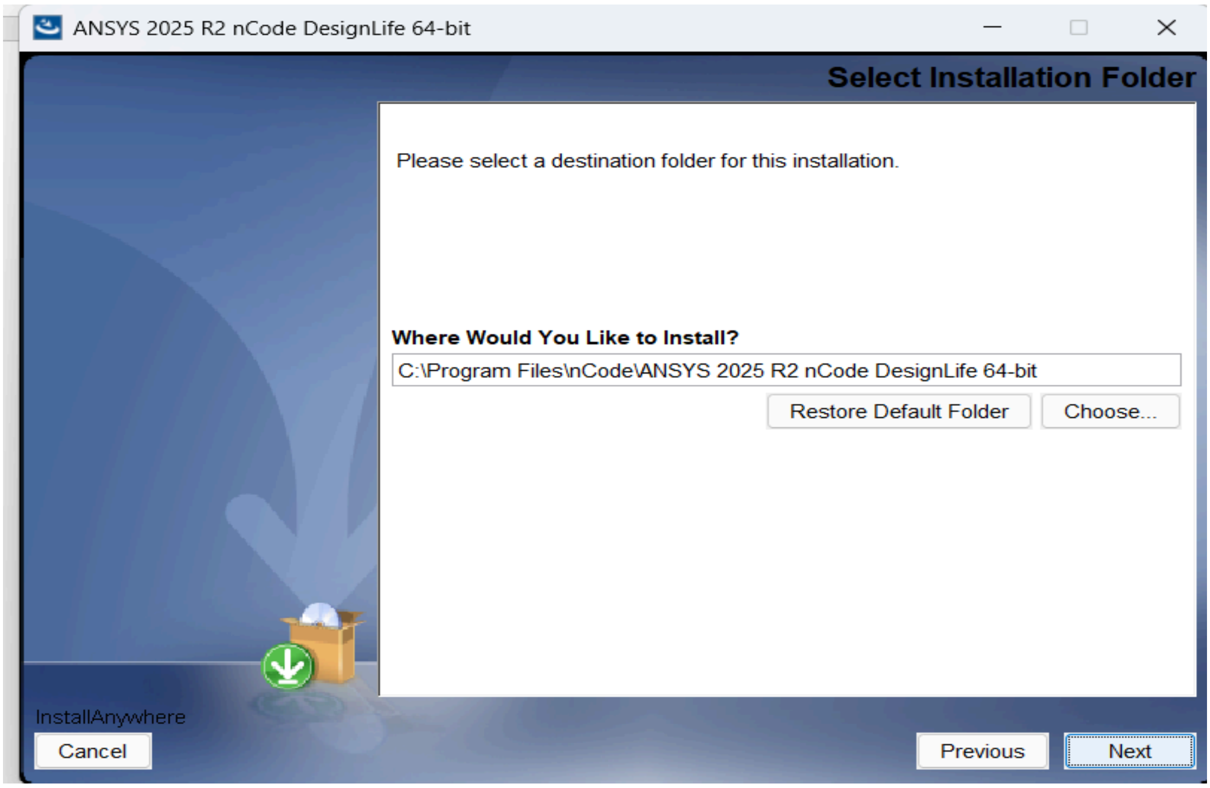
Step 2: Click Next



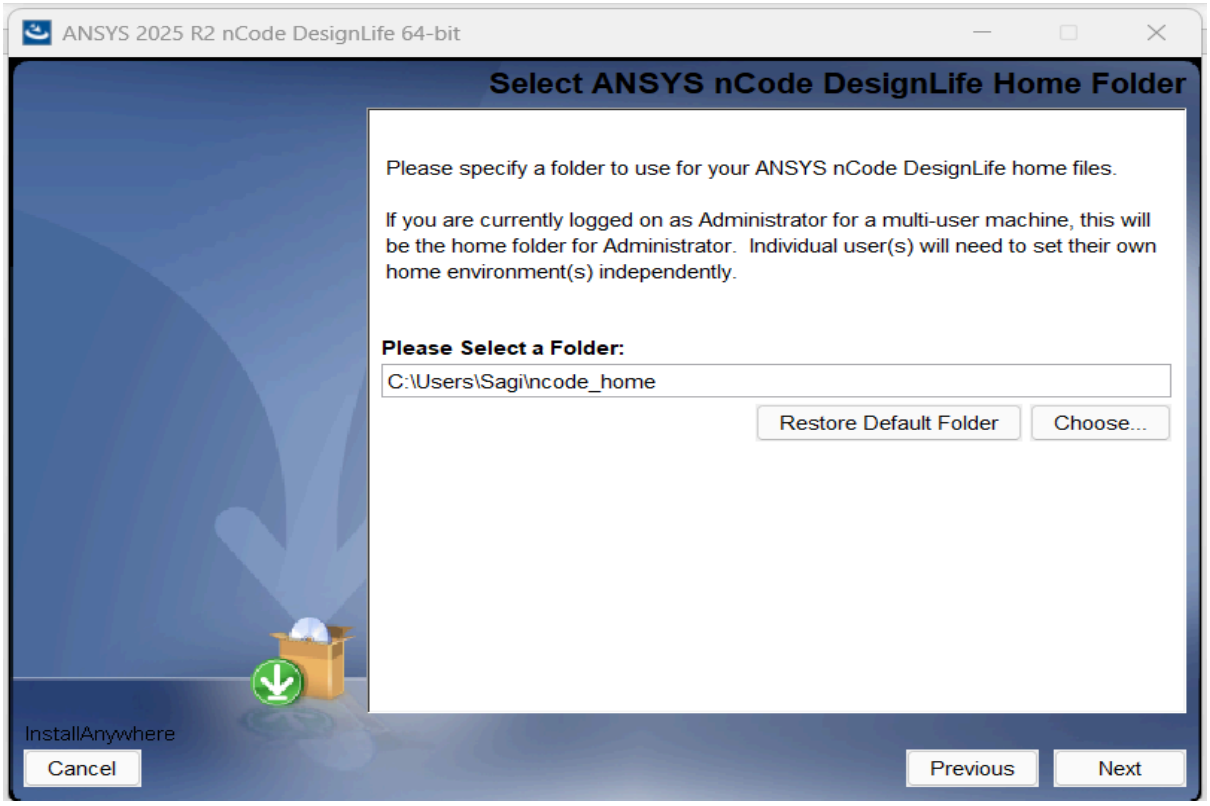
Step 3:Click on “I accept the terms of the license agreement”



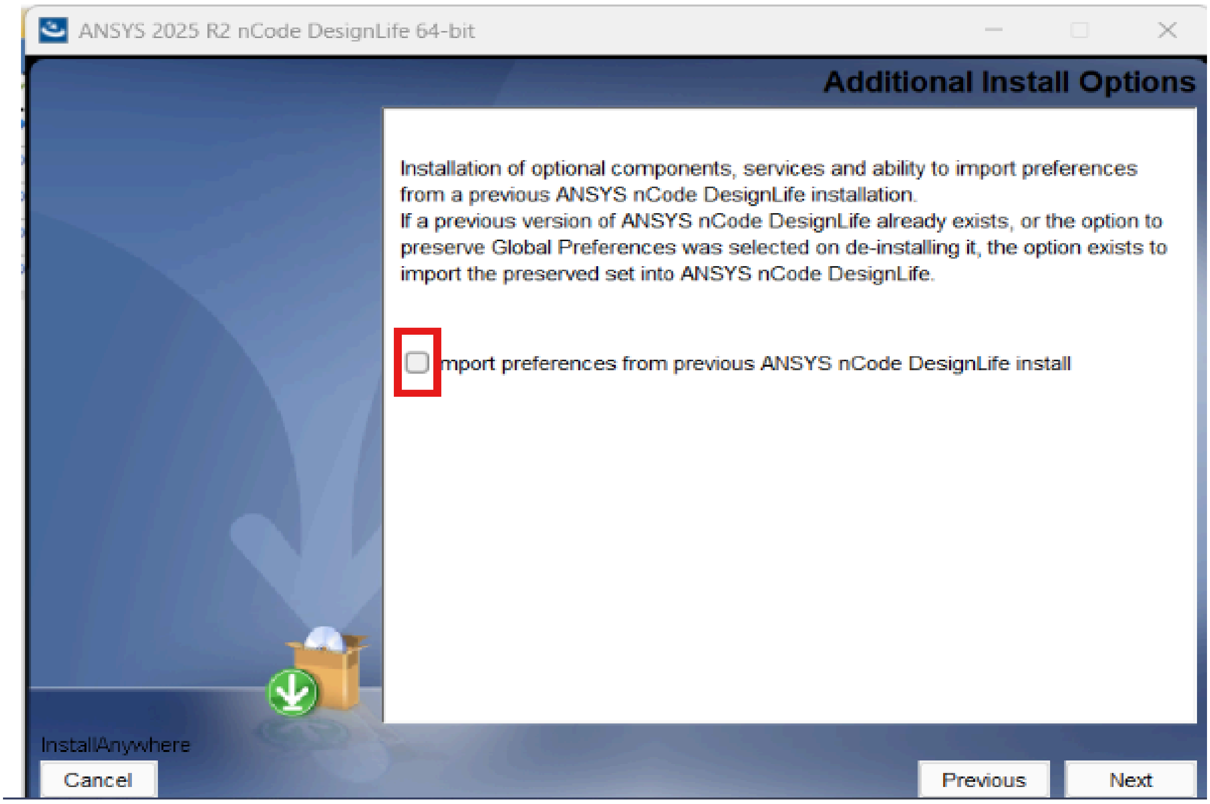
Step 4: You can choose the default path or install it to a different path if needed



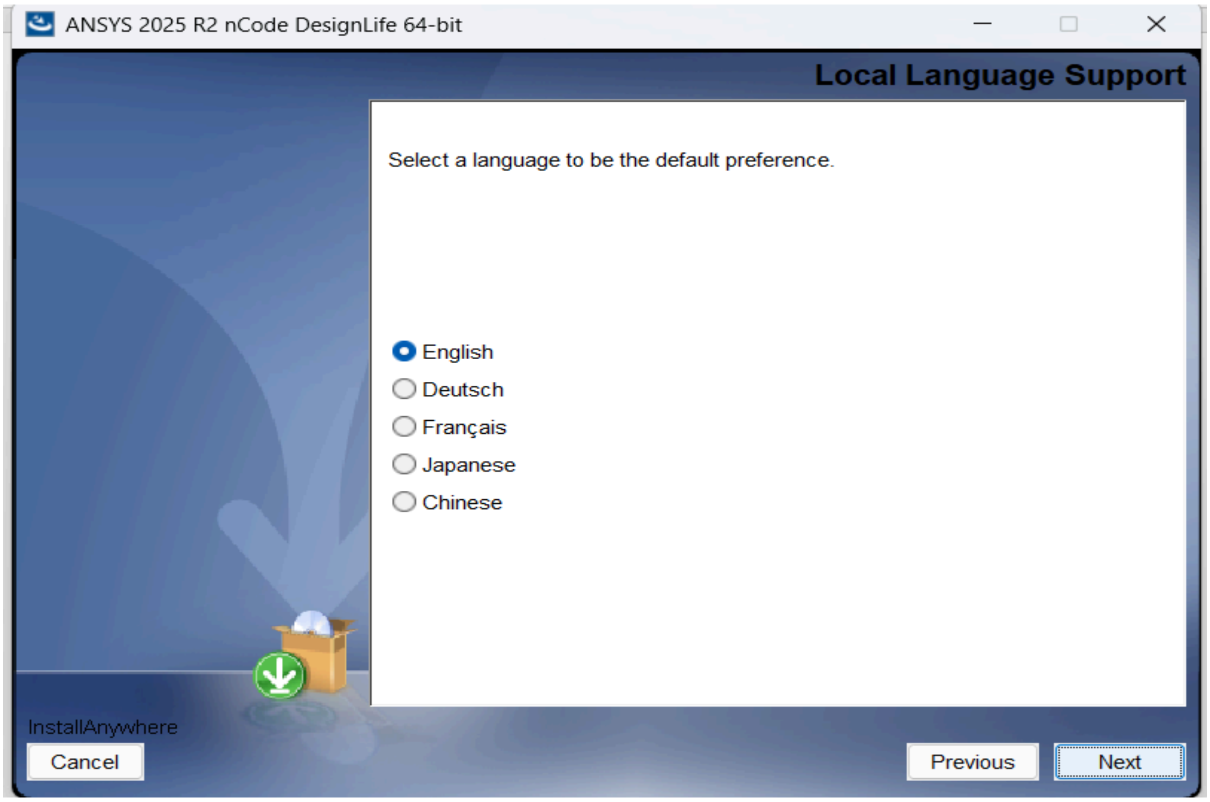
Step 5: Select the path for your nCode DesignLife home files



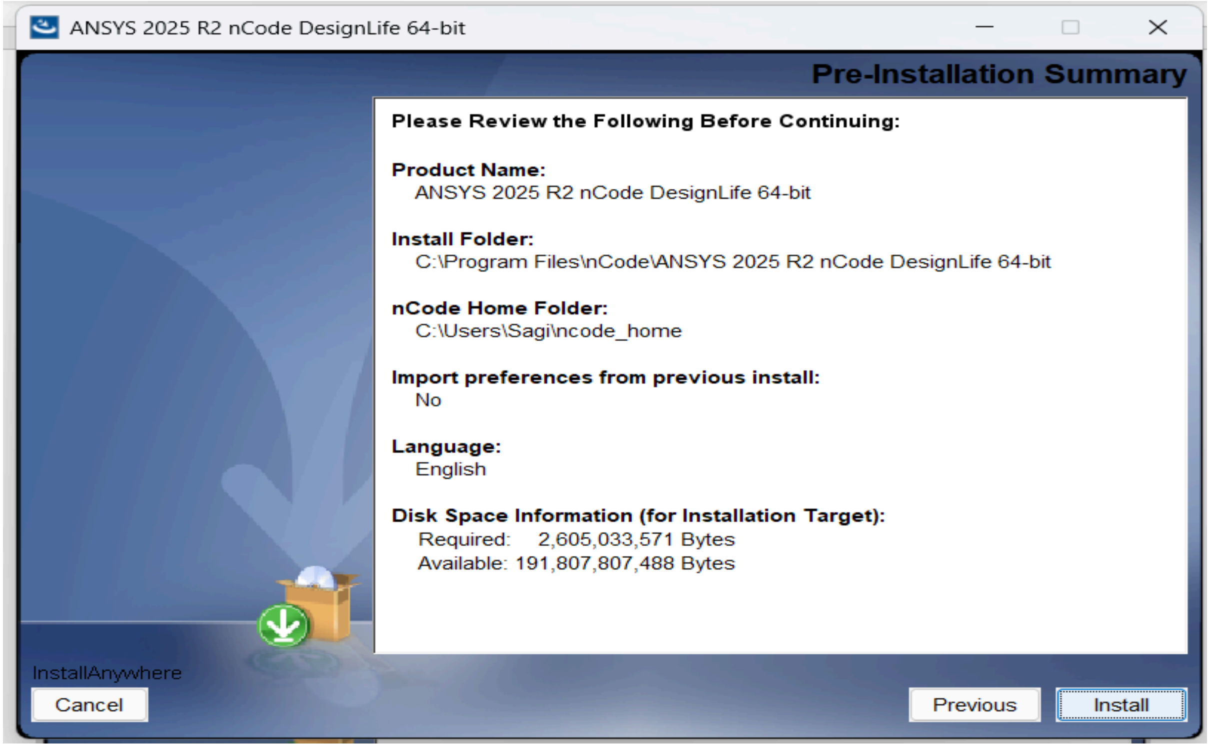
Step 6: Leave the checkbox unchecked



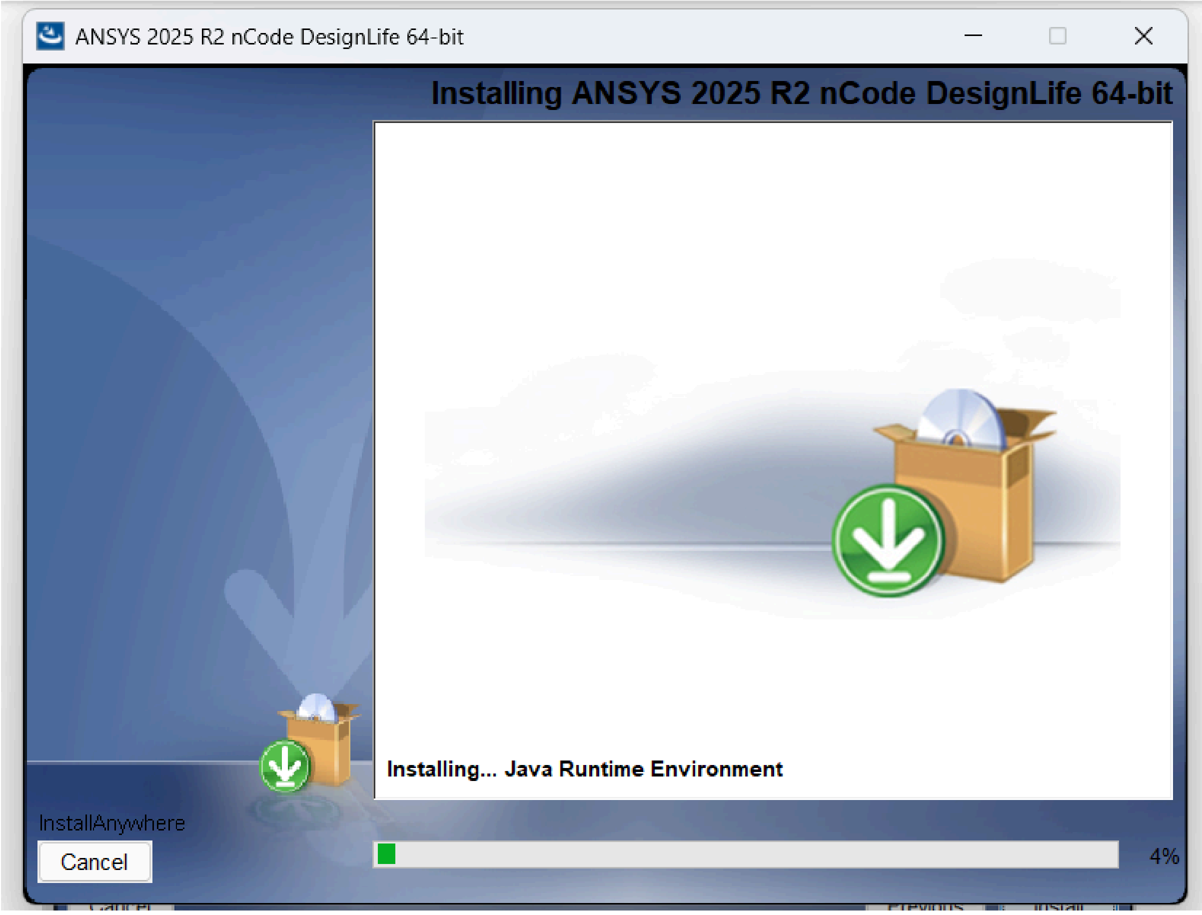
Step 7: Choose the Language



Step 8:Click “Install”



The installation has started. Wait for it to finish



How to install Ansys 2018R2 in Windows

To install the Ansys 2018R2, please refer this link

https://drive.google.com/file/d/1JcI_WfFzf1Oq6QG1Oq1TSavHDTB6kur9/view

Support/Help

In case of any issues, please raise a support ticket through
https://support.iitpkd.ac.in/