1. Navigate to the directory where you extracted the files and right-click the **setup.exe** file. Select '**run as administrator**'. The ANSYS Installation Launcher should appear.
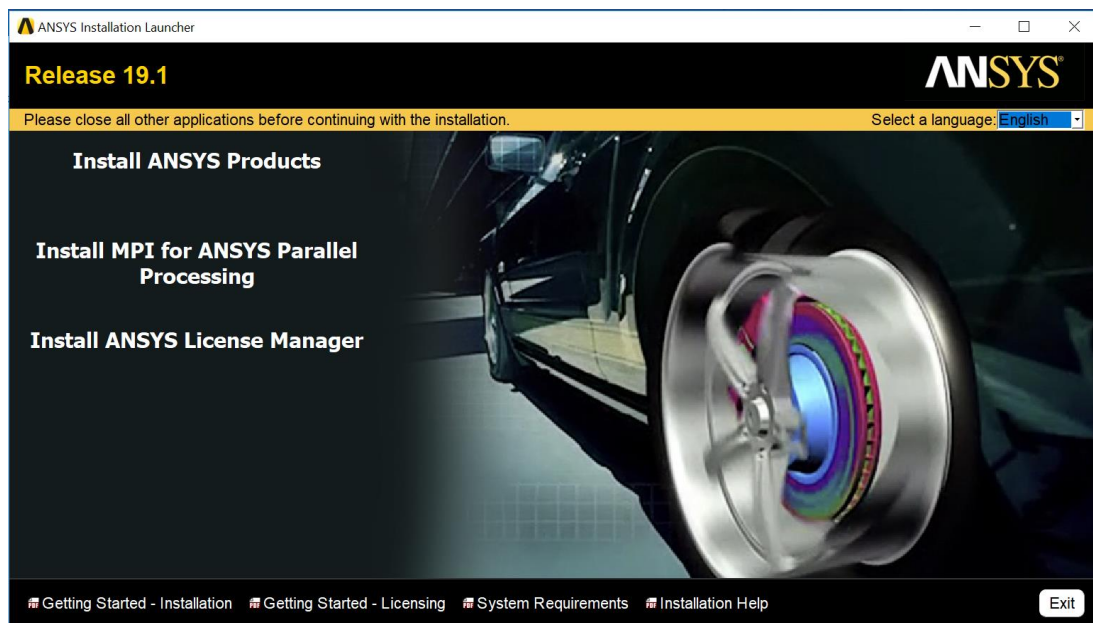

2. Click on '**Install ANSYS Products**'
3. The license agreement appears. Read the agreement and if you agree to the terms and conditions click '**I Agree**'. Click '**Next**'.
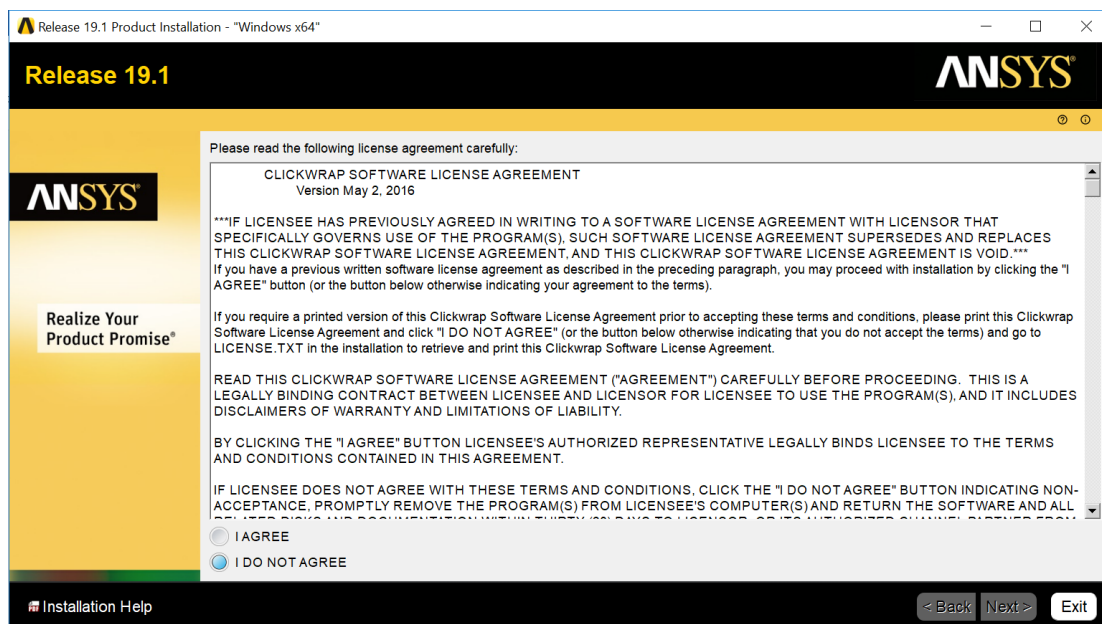
4. The directory where ANSYS Inc. products will be installed is shown in the **Install Directory** field. You can install the products into any directory you wish (ensure you have write permissions to use the directory you choose). The default (and recommended) directory is C:\Program Files\ANSYS Inc
Leave the **Associate files extensions with ANSYS products** checkbox marked. This allows you to double-click files from Windows Explorer to launch the various products. Click '**Next**'.
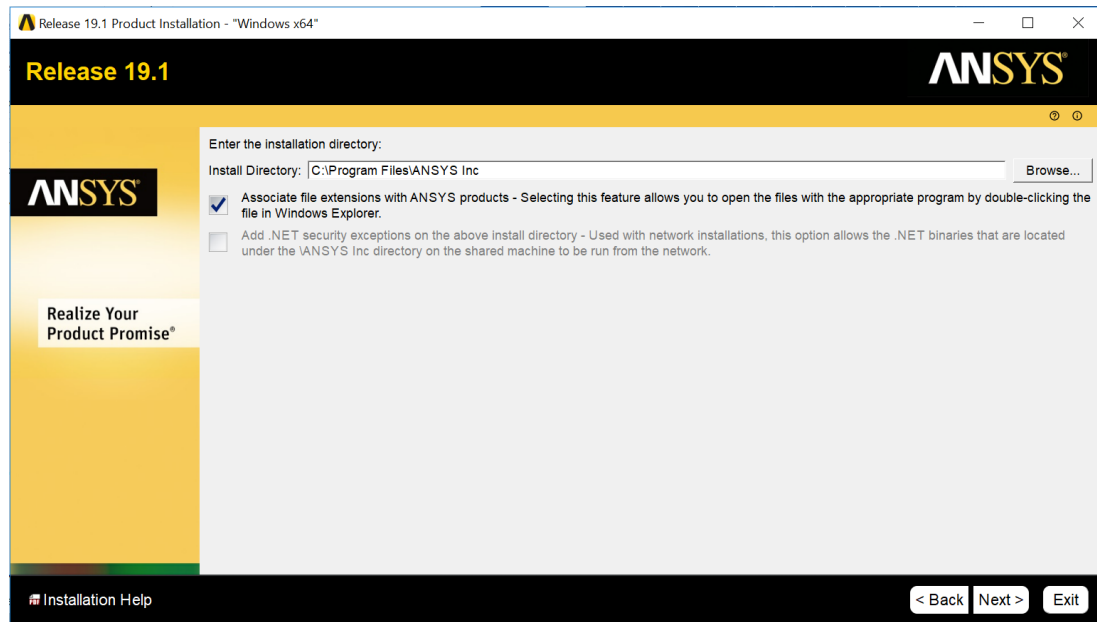

5. All products available in the installation package(s) you downloaded are listed. Select or deselect any combination of products. There is no individual product selection for ANSYS Workbench as it is automatically installed with most ANSYS Inc. products. Note: If you select AutoCAD, Catia V5, Inventor, NX, Creo Parametric, Solid Edge, SolidWorks, ICEM CFD, Creo Parametric, or ICEM CFD NX, you will have additional installation steps (not covered in this guide). Select/deselect the products you want to install and click '**Next'**.
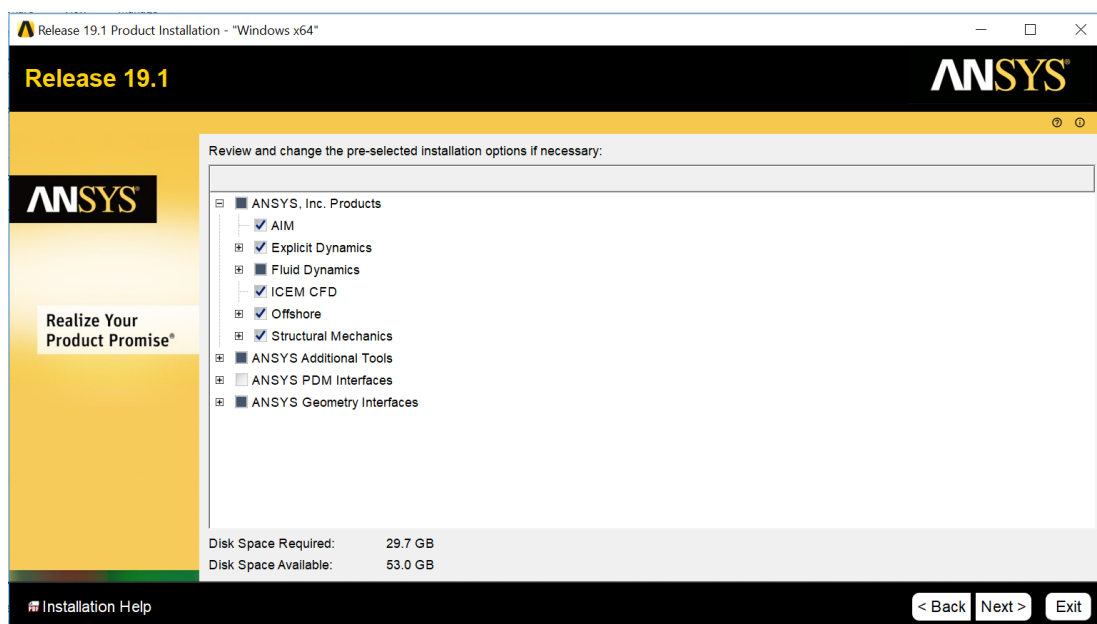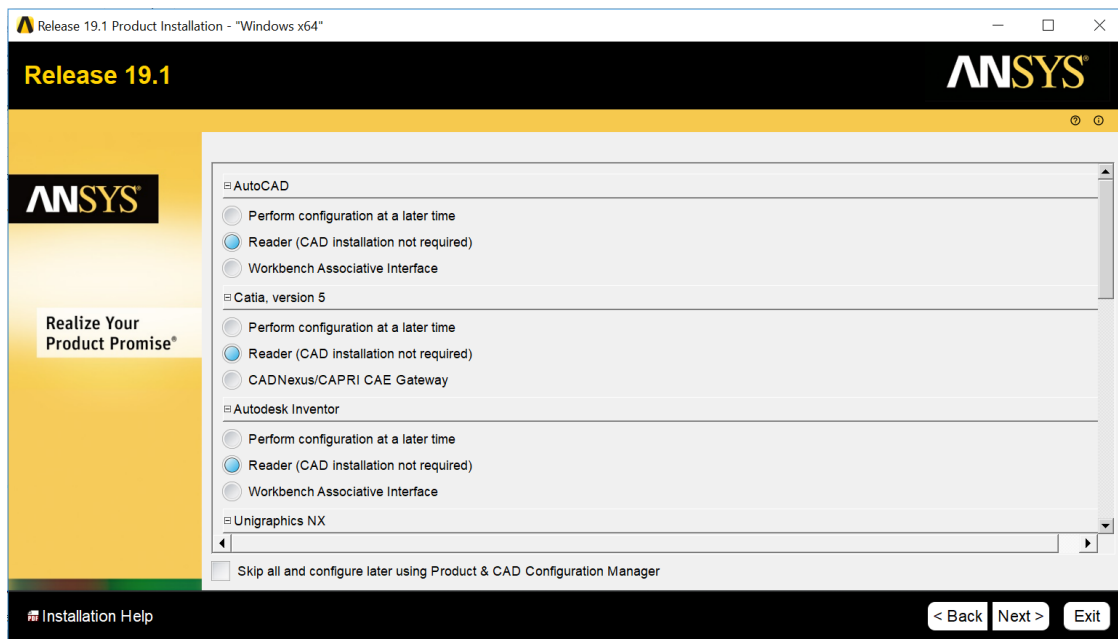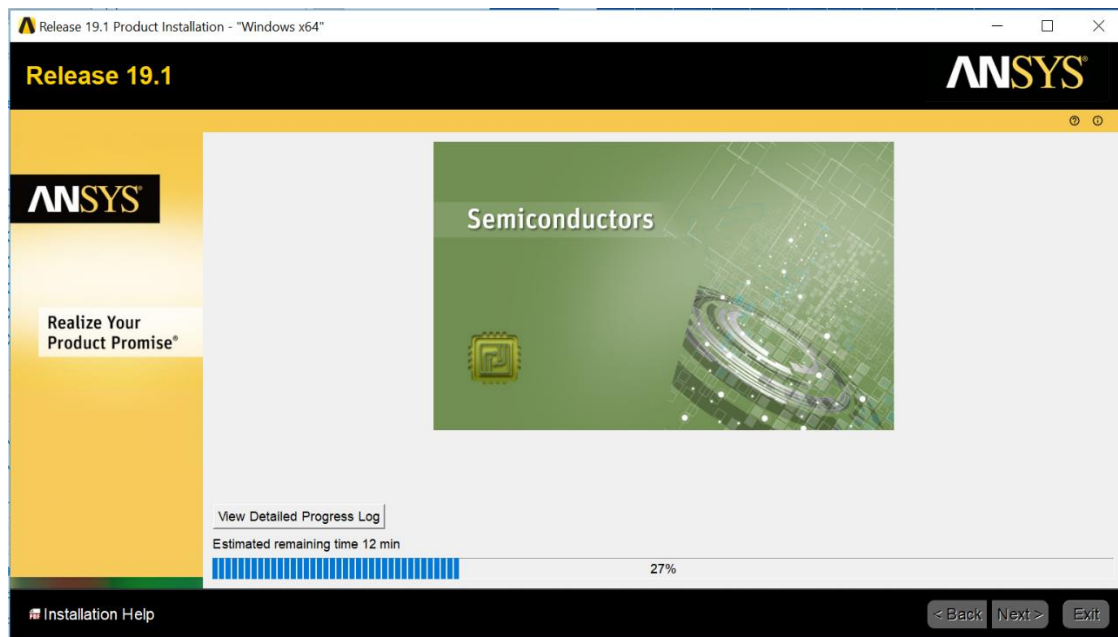
6. Click '**Next**' and you should reach a summary screen that lists everything you have chosen to install. Review this list to ensure it is correct. If it is not correct, go '**Back**' to the earlier screens and make the correct product selections.
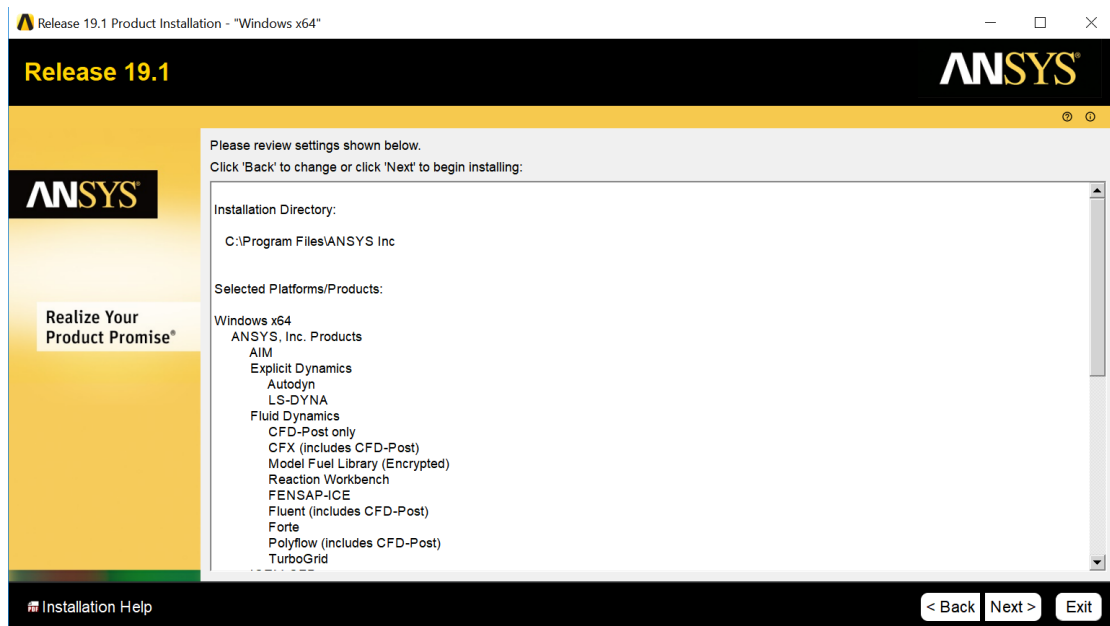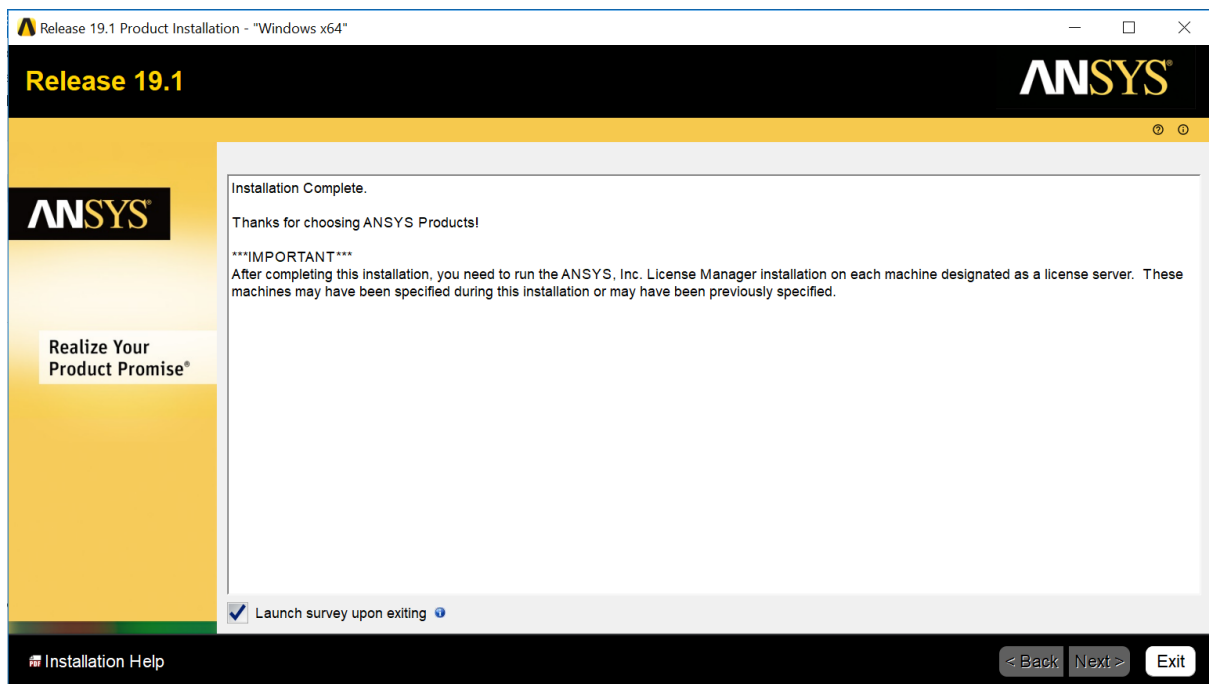

If the listed products are correct, then click '**Next**'.

7. At around the 50% mark during the installation, a message as shown below will appear. Select the path to the Next Media and then click '**OK**'. The installation should then continue.
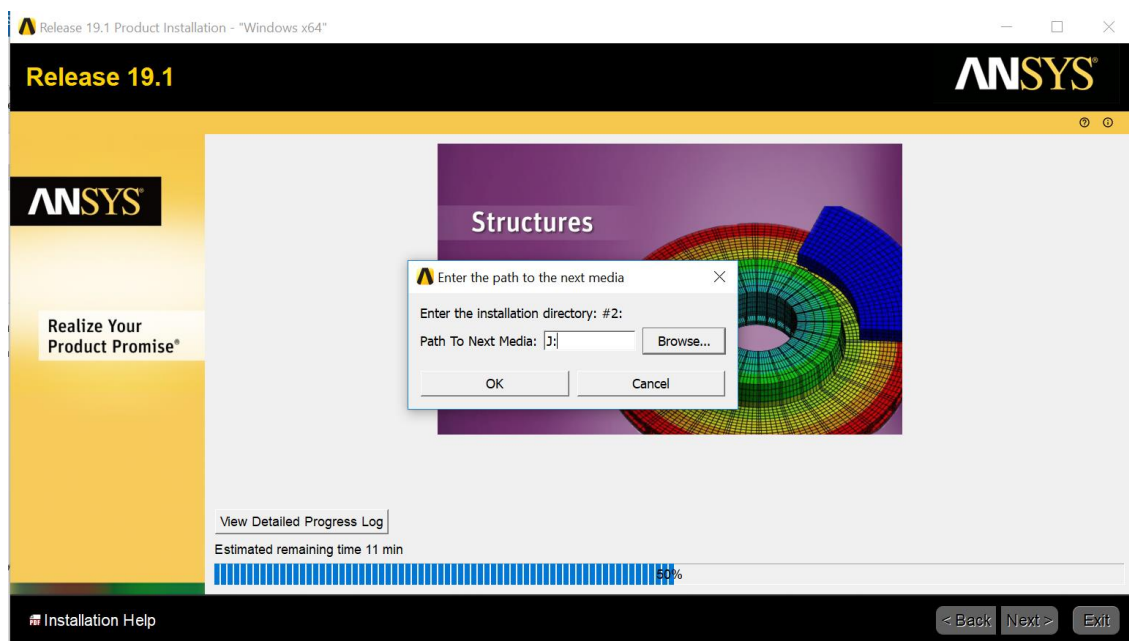
8. Once the installation is complete, the following screen should appear.

9. When ANSYS 19.1 is launched, it should appear as follows: