License Information

- 50 Task ANSYS Research Mechanical & CFD Bundle

- 50 Task ANSYS Research HF Bundle

- 50 Task ANSYS Research EM Bundle

- 500 Task ANSYS Academic Teaching Mechanical & CFD Bundle

- 500 Task ANSYS Academic Teaching HF Bundle

- 500 Task ANSYS Academic Teaching EM Bundle

- 5 Task ANSYS Academic Research CFD (Chemical Kinetics)

- 1 Task ANSYS Academic Research SCADE Bundle

- 25 Task ANSYS Academic Teaching SCADE Bundle

- 150 Task ANSYS Academic Space Claim Bundle (Mech)

- 256 Core ANSYS Academic Research HPC for FEA and CFD

- 256 Core ANSYS Academic Research HPC for HF and EM

- ANSYS Academic EnSight Post Processing Tool (5 Task)

- 5 Task ANSYS Research Mechanical including Additive suite bundle

- 5 Task ANSYS Discovery Simulation

- ANSYS Academic Lumerical Research–5 Task (Product with 3 Year TECS)

- ANSYS GRANTA Edupack – 10 Task (Product with 3 Year TECS)

- ANSYS LSTC University Department – 500 Core (Annual Lease for 3 Year)

- 5 ALH Licenses for 1 year

ANSYS Installation Procedure

After completing the software download, run the setup from Disc-1.

During the installation process, when prompted for the hostname, enter 10.2.60.16 and continue with the default port numbers.

When prompted, browse for the Disc-2.

Additionally, if you wish to install the Electronics tool, it can be found in the Electronics folder.

If you are unable to connect to the license server after installation, please download the "firewall.bat" file. Run it as an administrator and allow it to run it anyway.

This should resolve the issue and allow Ansys to run successfully.

Sometimes, users complete the installation process using their local machine name. In such cases, they should navigate to C:\Program Files\ANSYS Inc\shared\licenses\ansyslmd.lic and open it in Notepad as an administrator. They need to edit the file and replace the local machine name with 10.2.60.16.