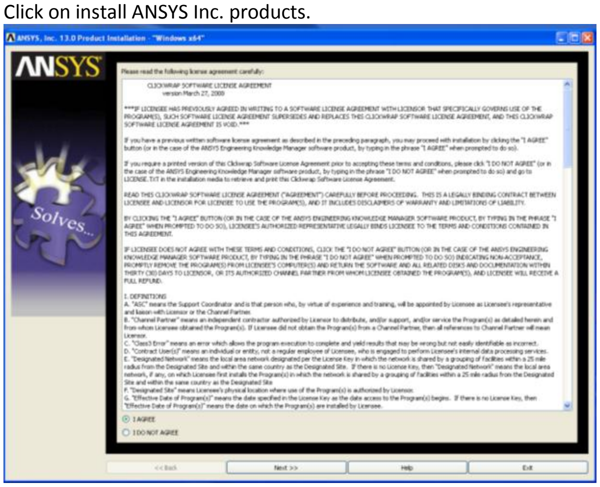
[ANSYS 13.0](#)

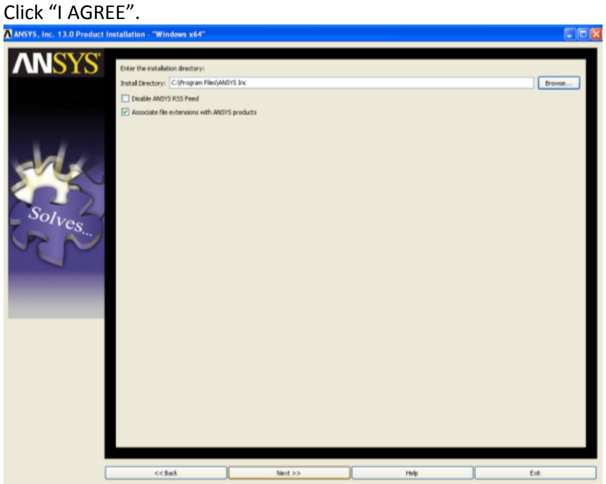
SERC has installed Ansys 13.0 on sunlx1_10. For details see the Ansys software page. The DVDs for the windows version are available in SERC Library (room no. 103). The Procedure for installation is given below.

There are two DVD's. Insert DVD-1.

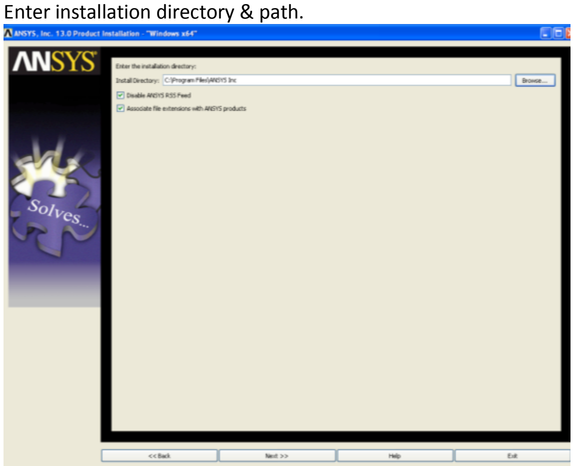
Click on install ANSYS Inc. products.



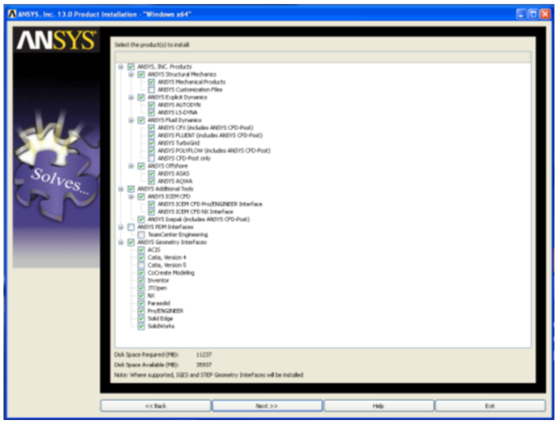
Click "I AGREE".



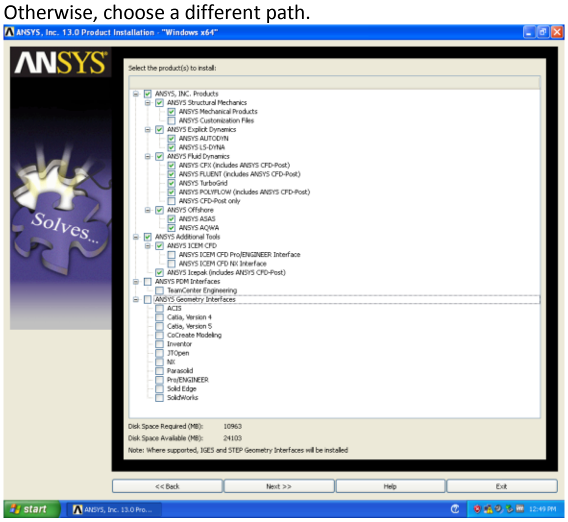
Enter installation directory & path.



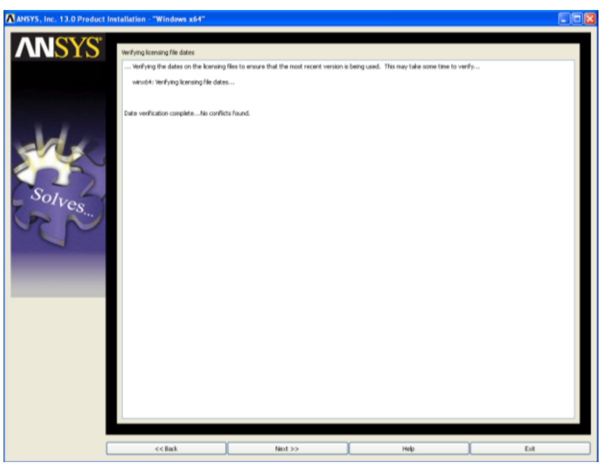
Tick 'Disable ANSYS RSS feed', click next.



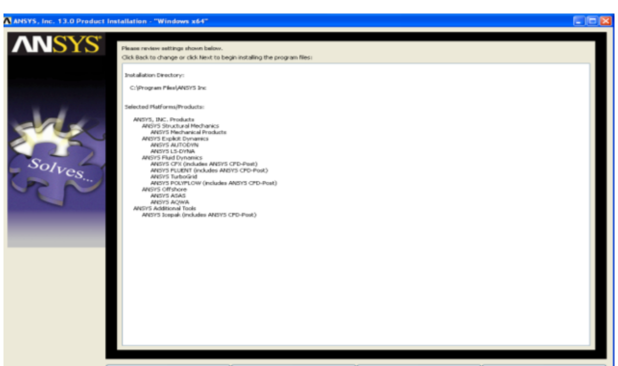
Disk availability is checked. If it is alright continue with the next steps. Otherwise, choose a different path.



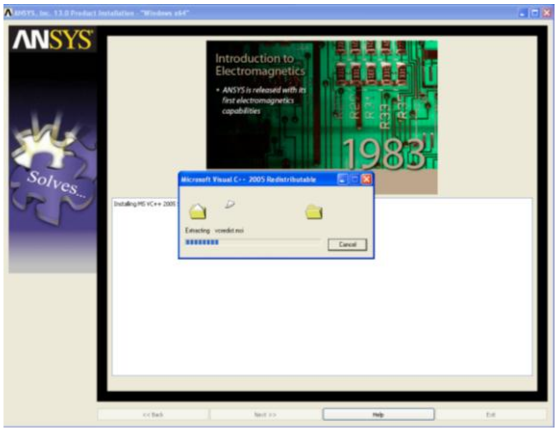
Uncheck the ANSYS ICEM CFD Pro/Engineer Interface, ANSYS ICEM CFD NX Interface and ANSYS Geometry Interface, then click next.



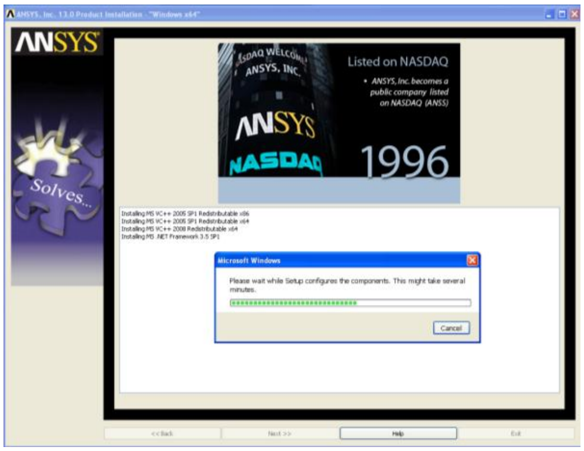
It will verify whether your system is 64 bit or not. Click next.

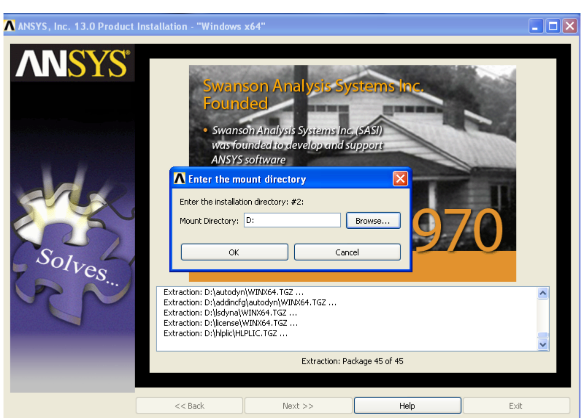


Click next for starting the installation.

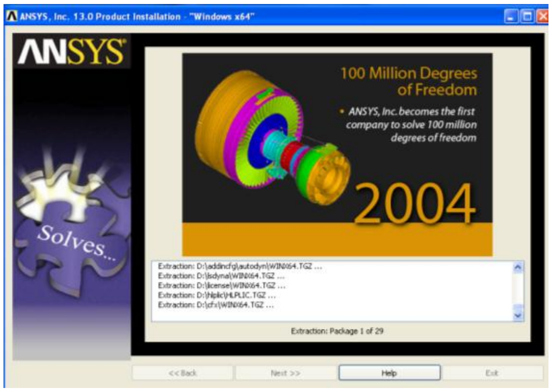


The installation begins

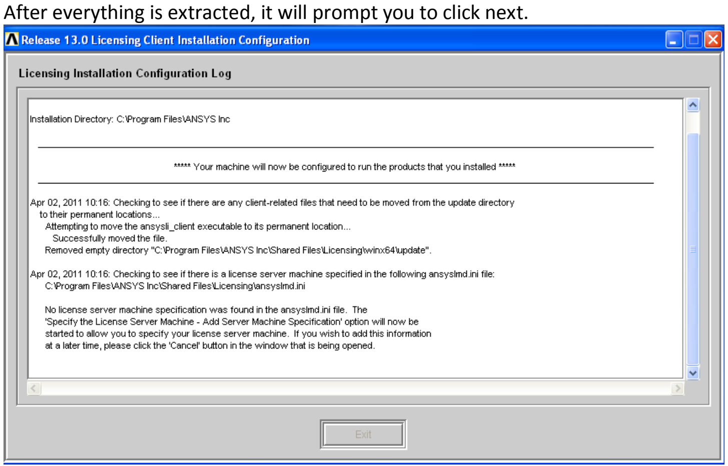


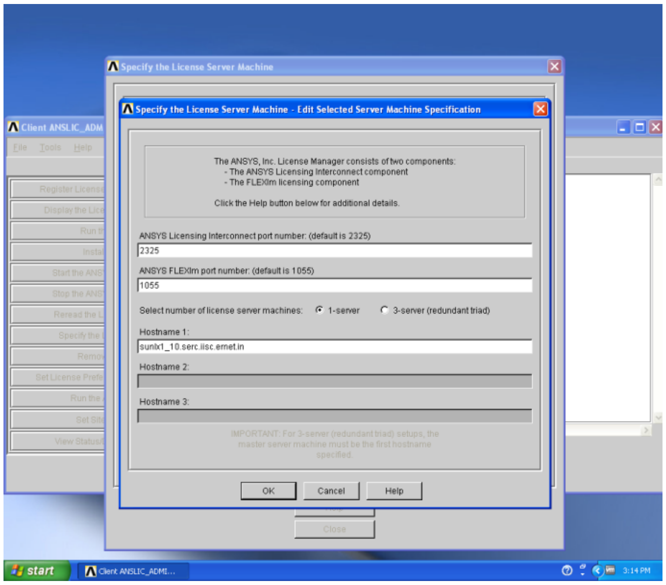


After extraction of all the packages in DVD -1, it will prompt for inserting DVD -2. Insert DVD-2 and click ok.

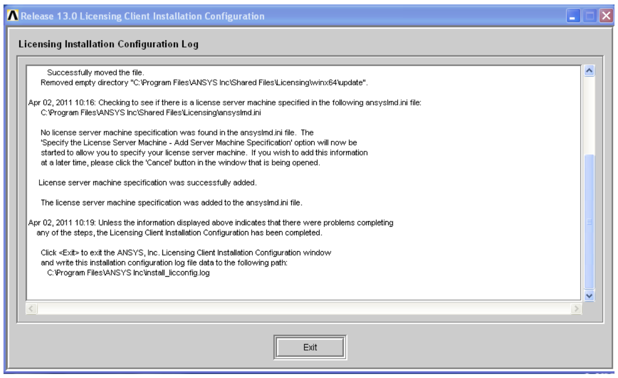


After everything is extracted, it will prompt you to click next.

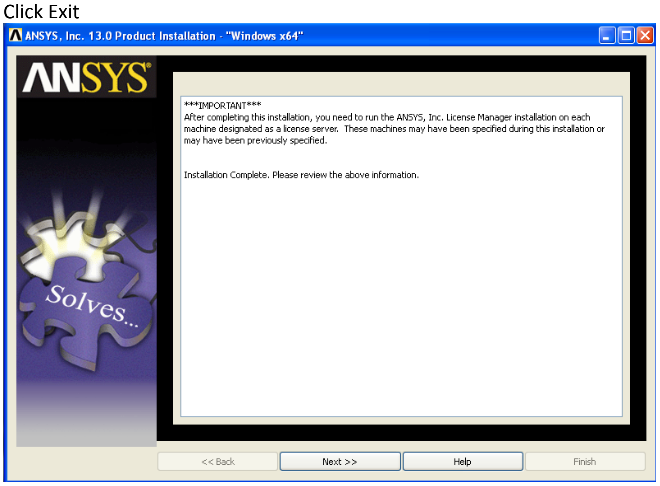




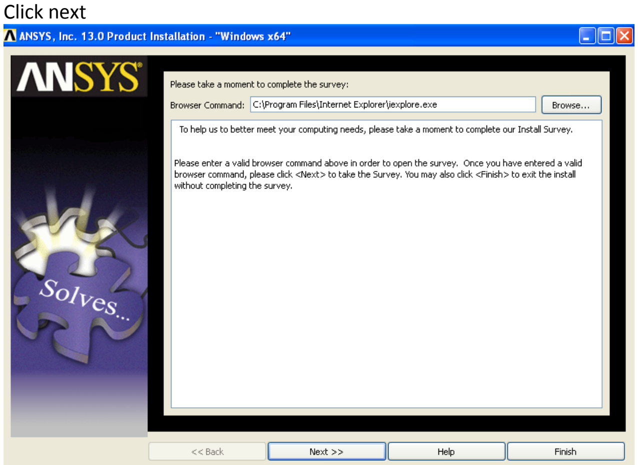
For Hostname 1 enter the license server's name :
sunlx1_10.serc.iisc.ernet.in and then click ok.



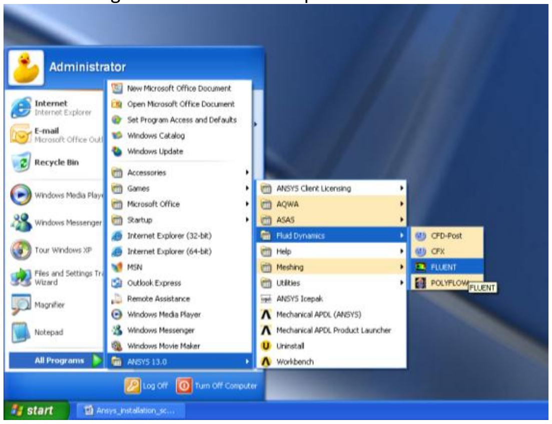
This shows that it has taken the proper server and license is available.
Click Exit



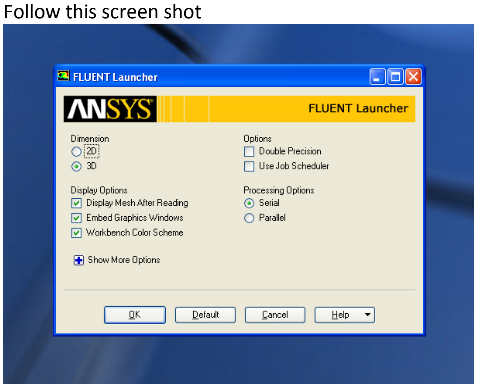
Click next



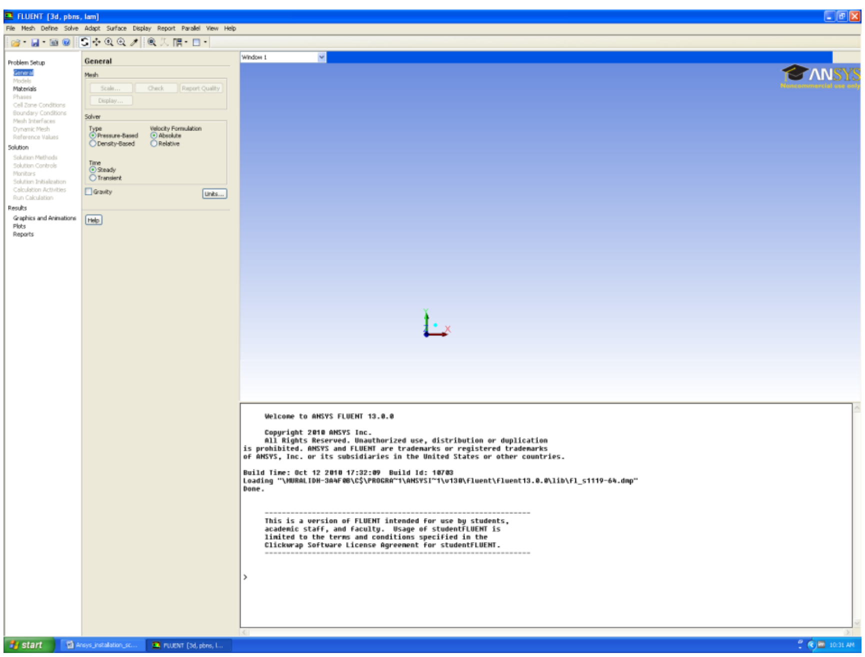
Click finish. Ansys has been successfully installed on your system. It is now ready for use.
To start using Fluent follow the steps below.



Now check whether it is installed correctly or not.
Follow this screen shot

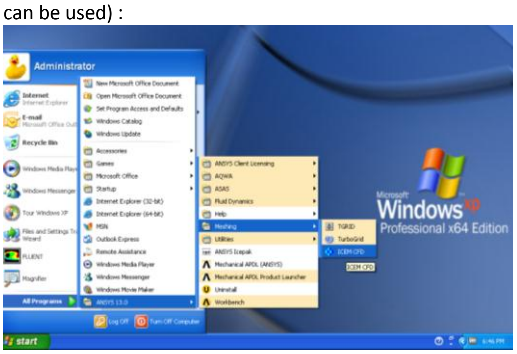


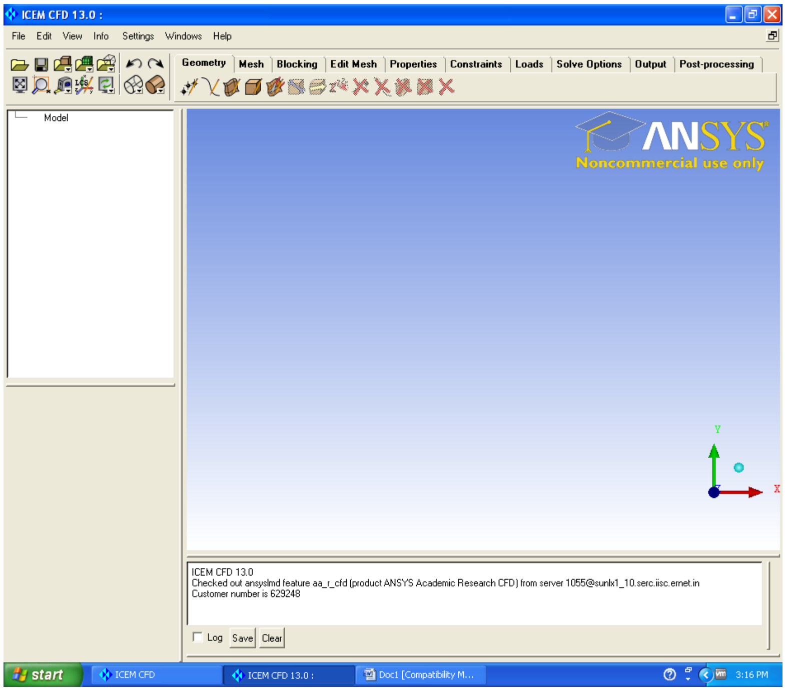
Tick 2D or 3D and click OK.



After clicking ok, this window will appear. Fluent is running.

To use ICEMCFD (this version does not have gambit, instead of that this can be used) :




ICEM CFD 13.0 :
File Edit View Info Settings Windows Help
Geometry Mesh Blocking Edit Mesh Properties Constraints Loads Solve Options Output Post-processing
Model
ANSYS
Noncommercial use only
ICEM CFD 13.0
Checked out ansyslmd feature aa_r_cfd (product ANSYS Academic Research CFD) from server 1055@sunlx1_10.serc.iisc.ernet.in
Customer number is 629248
Log Save Clear
start
ICEM CFD
ICEM CFD 13.0 :
3:16 PM