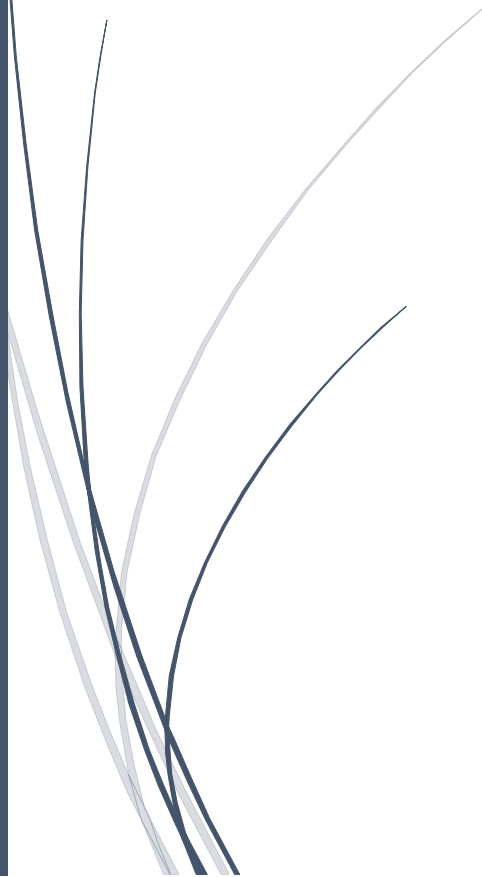# ANSYS LICENSNG FAQ

TABLE OF CONTENTS:

I.   Server down or not responsive

1. **I am getting the following error:** *"Could not connect to any license server, the server is down or not responsive".* **What do I do?**

   a. Make sure user have access to license server. You can test by pinging the server: In windows search, type "cmd", a command prompt window will open, type "ping *<server name>*", if the ping is successful you should see several 'Reply from…' lines. If it is not successful:

      i. Make sure the server machine is up and running
      ii. Make sure that the ANSYS Licensing Interconnect port number, ANSYS FLexlm port number, and server Hostname are inputted correctly:
      To check and correct them, run the client Anslic_admin utility **as administrator,** click on "Specify the License Server Machine", select the server machine and click on "Edit Selected Server Machine", verify the fields are inputted correctly (should match the license and the server). See photo below:
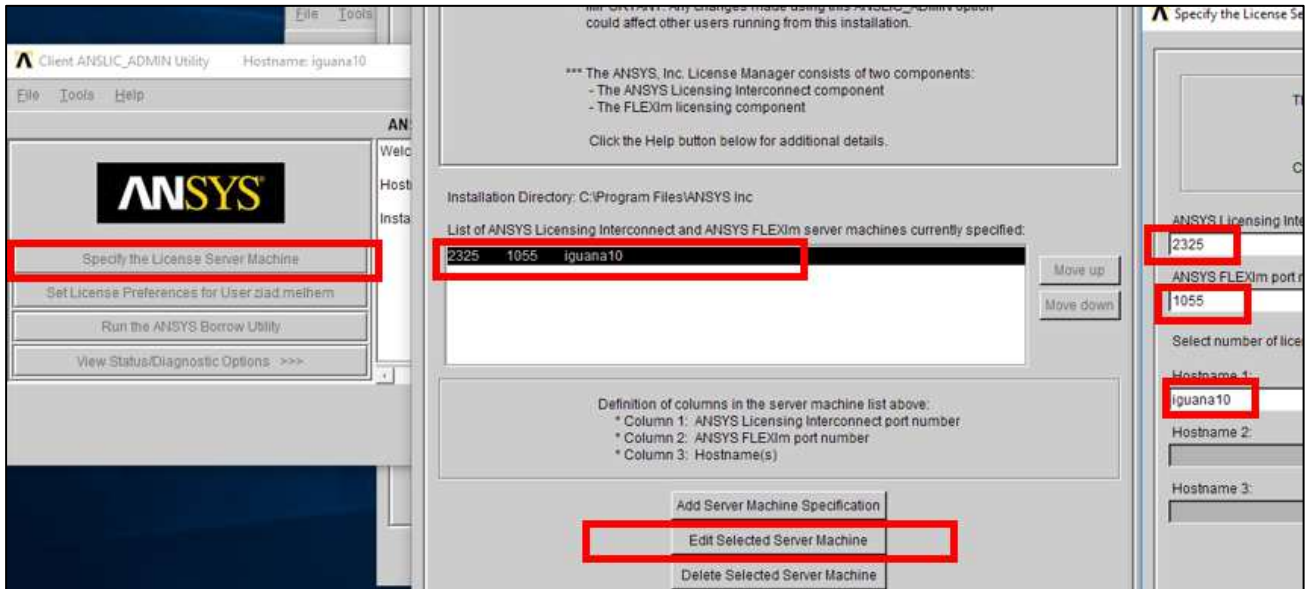
      iii. Make sure you have access to the specified port at the server.
      You can check if a port is open by using the telnet command. First you will need to activate it (it is turned off by default):
      Start-> Control Panel-> Programs-> Turn Windows features on or off and check the box next to Telnet Client

Once it is activated, Type cmd in windows search, a command prompt window will open. Type telnet [domainname or ip] [port] If the port is open, you will see a blank screen. If the port is closed, you will receive a Connect failed message. If you do not have access to license server through the specified port, please contact your IT department.
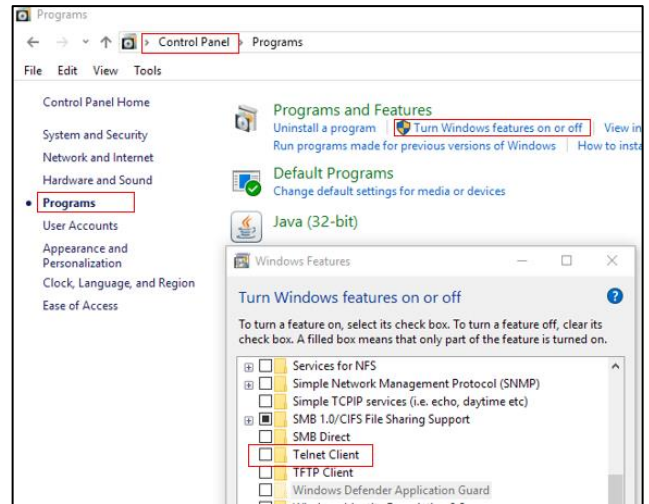
b. If there is a firewall (for detailed info check 7.4 of the licensing help guide)
   i. Make sure certain ports (1055 2325 by default) can pass through it
   ii. Make sure ANSYS license manager firewall exceptions for license related executables are in place:
   (Files to add are found in the directory:
   C:\Program Files\ANSYS Inc\Shared Files\Licensing\winx64
      SERVER-side Exceptions)
      1. lmgrd.exe
      2. ansyslmd.exe
      3. ansysli_server.exe
      4. ansysli_monitor.exe
      5. ansysli_client.exe

      CLIENT-side Exception:
      1. ansysli_client.exe

c. Especially after updating ANSYS on client, make sure server is updated as well. Server needs to be same or newer version: Inside the ANSLIC utility (client or server) Help -> About ANSYS Inc Licensing can be used to check the version number
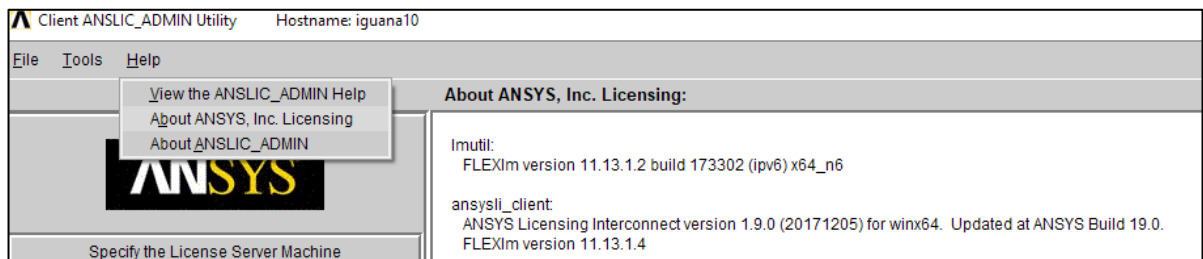
d. ANSYS does not currently support IPv6, make sure that the network is not only using IPv6

e. Try restarting the client and the server (first make sure no other users have active runs)
f. Fix incomplete installation: some versions of the Client have a menu option: Tools->Complete Unfinished Licensing Installation Configuration (requires admin rights)
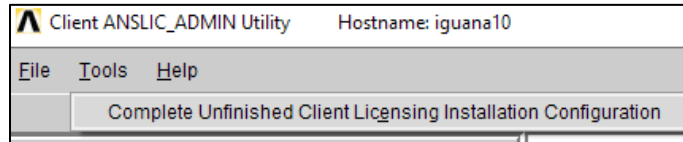
II. Installation/Migration

1. **How do I register for the ANSYS Customer Portal?**

   Please follow the instructions in the following video:
   https://youtu.be/M0mQGjaVBxE

2. **How do I download ANSYS from the ANSYS Customer Portal?**

   Please follow the instructions in the following video:
   https://youtu.be/FmQendBsEO8

3. **How do I install ANSYS and the ANSYS Electromagnetics Suite on Windows?**

   Please follow the instructions in the following video:
   https://youtu.be/QxS0kWOK-bw

4. **How do I download and install the ANSYS license Manager on Windows?**

   Please follow the instructions in the following video:
   https://youtu.be/h46h4_DVftA

5. **How do I configure a Client Machine to point to the ANSYS License Server?**

   Please follow the instructions in the following video:
   https://youtu.be/FcluoQPw7iA

6. **How do I configure ANSYS Electromagnetics to point to the ANSYS License Server?**

   Please follow the instructions in the following video:
   https://youtu.be/vwVlvlF4v6w

7.   How do I add or replace ANSYS Licenses on Windows?

Please follow the instructions in the following video:
https://youtu.be/i4C4MC3h59c

8.   If the Mac address of the license server change, do we need a new license?

Yes

9.   I am going to move our license server to a different server. I believe I will need a new license, what information do you need from me to generate a new license?

You are able to transfer the license to a new license server yourself if you are the Account Support Coordinator (ASC). First, you will need to obtain the hostname and FLEXID for the new computer.

Within the ANSYS License Management Center on the new license server machine once the ANSYS License Manager has been installed, there is a button to obtain the System Hostid Info, as shown in the photo here:

Once you have that hostname, etc. information, you can go back to the ANSYS Customer Portal and request a license server change. We created a video (requires a password) that shows how to do it. Here is a link to the video as well as the password to view it:

https://vimeo.com/131654992
Password: padtlc

10. **I obtained an additional license and I already have ANSYS installed, how do I add it?**

Run the License Management Center on the server and then pick the Add a License File option. It will then prompt for the new license file and you need to point it to the text file that was e-mailed to you.

III.   VPN

1. **I am using VPN to remotely access my work computer. I am able to run work sites but not ANSYS. How can I fix that?**

   a. Make sure the firewall exceptions are set (see I.1.b. in this FAQ)
   b. Use the IP address on the client machine instead of the machine name to deal with any DNS issues that may arise from using a VPN connection.
   c. Add a system level environment variable to extend the time the client software waits for a response from the server: Right Click My Computer -> Properties - Advanced System Settings -> Environment Variables. Click on the bottom "new" button: Variable Name: FLEXLM_TIMEOUT, Variable Value: 15000000. Reboot the machine to make sure the setting takes effect
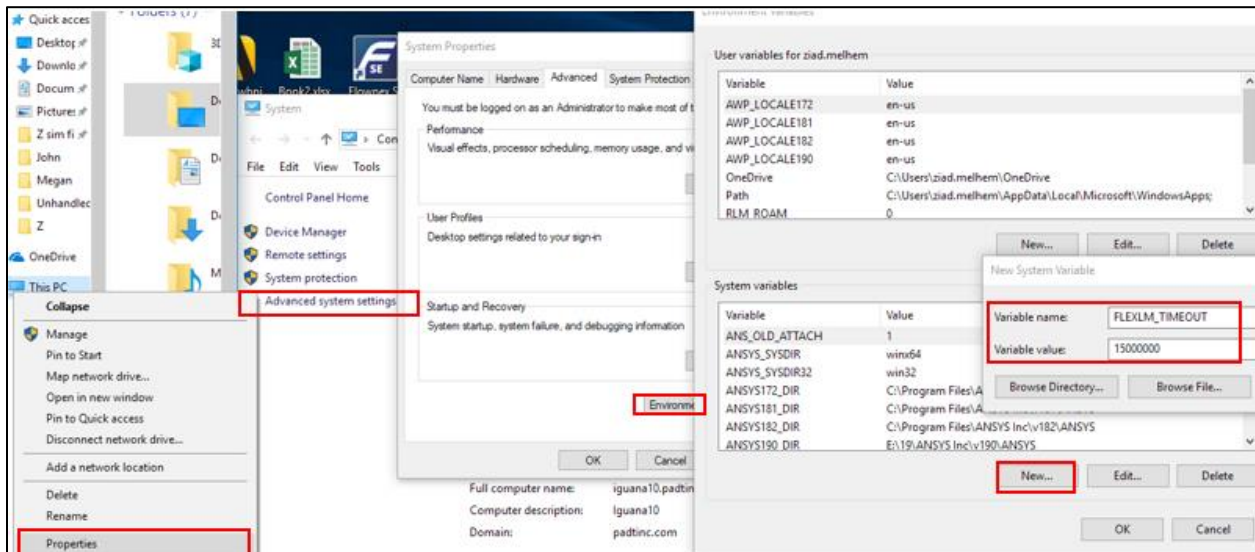
IV. TECS and license expiry

1. **My license indicates that it is permanent with TECS expiring on a certain date. What does that mean?**

   That means the license is permanent, but the technical enhancements and support (TECS) expire on the date indicated.

2. **How do I know if I can have access to the new release of ANSYS with my current license?**

   You will have access to the new release if the TECS (technical enhancements and support) end date is later than the build date of the new release.

V. Versions compatibility

1. **I am getting the following error message: *"Your version of the ANSYS license manager software is out of date. Please download and install the current ANSYS license manager form the ANSYS Customer Portal. The version of the license server ... [X.X.X] must be greater of equal to the client version [X.X.X]."* What do I do?**

   Please install the updated ANSYS license manager version on the license server. Note that if you are downloading a point release, e.g. 18.1 and have the 18.0 license manager you do NOT need to download the license manager.

2. **Is the license file/ license manager backwards compatible?**

   Yes. The most recent license file and the latest license manager will allow for running all prior versions going back to release 5.7.

3. **I have a new license file; do I need the latest license manager?**

   No, this is not required, but if you are running an older license manager it is recommended to always be running the most recent version of ANSYS License Manager. Please refer to ANSYS Platform Support page at http://www.ansys.com/en/Solutions/Solutions-by-Role/IT-Professionals/Platform-Support to make sure the most recent ANSYS License Manager is supported on the operating system of the License Manager machine.

VI.   Overuse of licenses

1. **I am getting an error stating that the:** *"Licensed Number of users already reached",* **is there a workaround it?**

   You don't have any licenses available because the licenses are all checked out. You can check with other users if they are actively using it or have left ANSYS open in the background. To determine who the users are, please open the Client ANSLIC_Admin Utility and click on Display the license status under View Status/Diagnostics options as shown in the photo below:
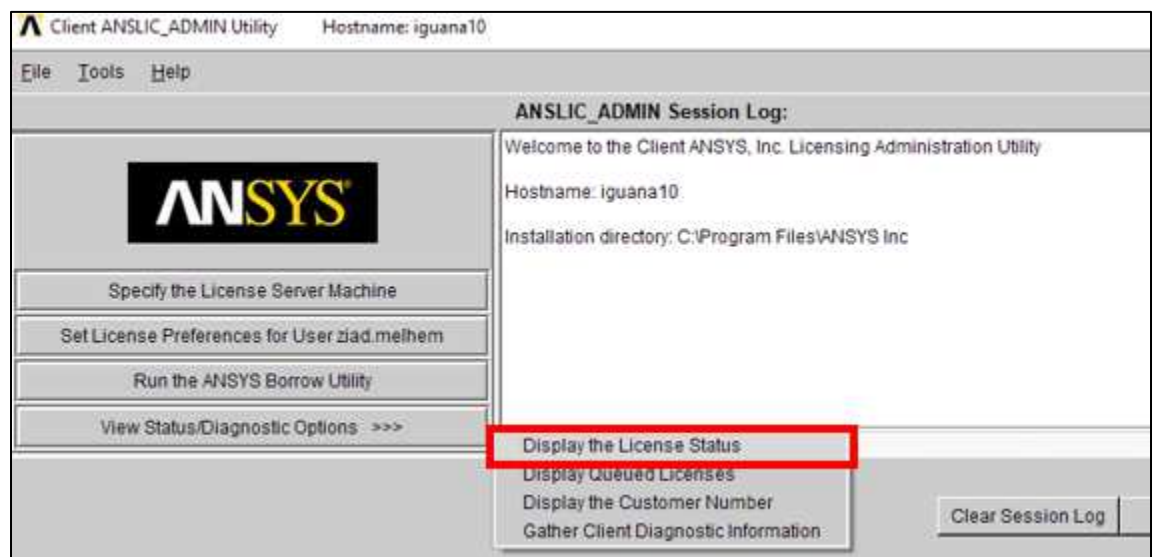
VII.   Include list

1. **I am getting an error message stating the user/host is not on the INCLUDE list, what does that mean?**

   This means the license server administrator has restricted access to the license (by creating a set of rules in a Flexlm options file). Please contact your license admin team to check if you can be added to the INCLUDE list.

2. **I am submitting a job using RSM, I get an error saying user/host is not on the include list. How do I fix that?**

   When RSM submits a job it may does so using a generic "SYSTEM" user which might not have permission to access the ANSYS licenses.

Go to Tools-->Options-->General Options-->Remote Solve

You can change it to a specified user and by using the username/password/domain of the network you can make it so it's the user's name that gets submitted to ask for a license

VIII. HPC

1. How to use HPC? What is the difference between HPC and HPC Workgroup?

First, ANSYS HPC and ANSYS HPC Workgroup licenses are exactly the same. The difference in terminology is only related to how many are purchased at one time. They are all ANSYS HPC Licenses.

Each HPC unlocks one compute core. HPC pack unlocks a group of cores (10 in 18.0). The use of ANSYS HPC vs. ANSYS HPC Pack licenses is determined via setting ANSYS Client License Preferences. This is accomplished in the Workbench window via Tools > License Preferences or from the Windows Start menu via Start > ANSYS 18.0 (or other version) > Client ANSLIC_ADMIN Utility (Run as Administrator) > Set License Preferences for User> then specify ANSYS Version Number (usually pick highest version) …Move the license you want to use to the top of the list in the HPC tab, then click OK (see photo below).

To actually utilize the HPC licenses during a solution, we put together an article showing how to do it for various ANSYS tools at this link:

http://www.padtinc.com/blog/the-focus/activating-hyperdrive-in-ansys-simulations
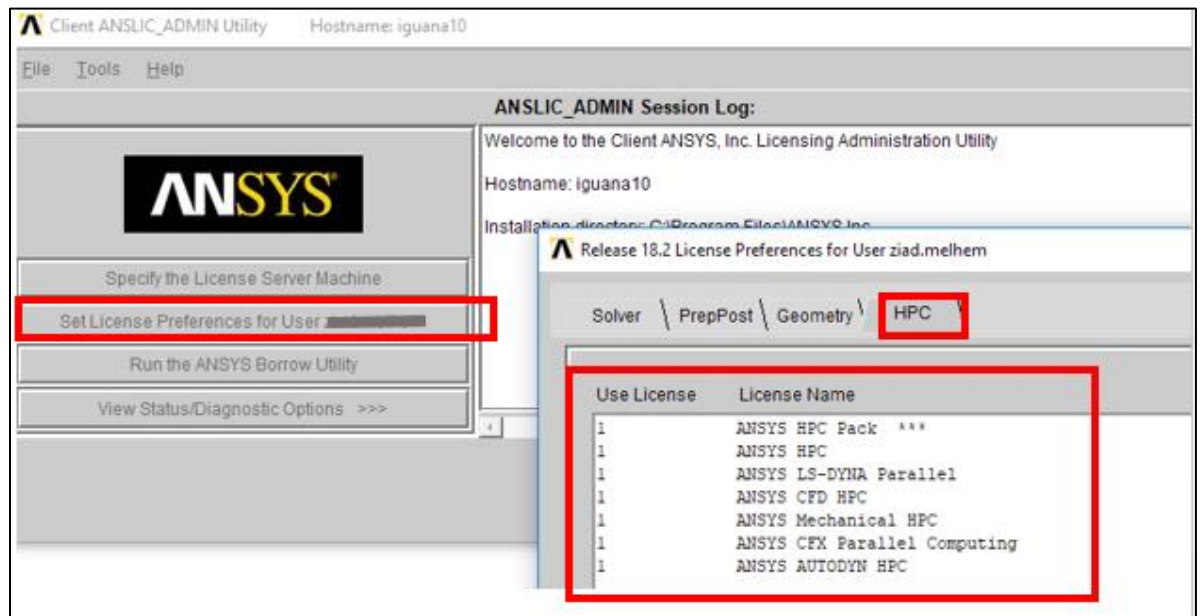
2. **When attempting to run a DCIR or Zo scan, the simulation fails with the error "Failed to check out license 'hfsshpc_pack'.", what do I do?**

The HPC licenses were recently changed to Pool instead of Pack. To change the license setting, click the Options button under the Simulation tab. Then, in the dialog box, select the Multiprocessing tab and set the HPC licensing to HPC Pool.

IX. Virtual server

1. **Is ANSYS supported on a virtual server?**

Yes