

How to Force Release of Ansys Licenses

On rare occasions, an Ansys license may fail to return when the software is closed. When that happens, the license can get 'stuck,' and the server will treat it as checked out even though it is not in use. The cause can be the software closing abnormally or a VPN connection being interrupted. It is recommended to first use the Task Manager to identify and terminate Ansys applications that did not close properly and may be occupying a license. If there are no hung processes, the next step is to manually release the licenses.

It is possible to force licenses to release from the client machine by identifying them with the client license utility, then removing them from the command line. First, run the **Ansys Client Licensing Settings** utility for the Ansys version you are using. Then click **FlexNet Publisher License Status** and find a line that corresponds to the license you wish to release. It will look something like the below:

Users of **INCREMENT**: (Total of 6 licenses issued; Total of 1 license in use)

"**INCREMENT**" v9999.9999, vendor: ansyslmd, expiry: 28-feb-2023
vendor_string: customer:00200105 tp:p
floating license

USER USER.DOMAIN.COM USERHOST.DOMAIN.COM DISPLAY# (v2021.1108) (**SERVER/PORT**
202), start Mon 5/16 11:16

Run a Command Prompt window. Issue the command below after making the appropriate substitutions based on the license status from the license utility. Underlined items will need to be edited, and the file path will need to be changed as well if using a non-standard installation location. In general the port and server will not change, so after you have used this once all you will have to change are the "INCREMENT" and "USER.DOMAIN.COM ..." fields, which can be copied directly from the license status display.

```
"C:\Program Files\ANSYS Inc\v221\licensingclient\winx64\lmutil.exe"  
lmremove -c PORT@SERVER INCREMENT USER USER.DOMAIN.COM  
USERHOST.DOMAIN.COM DISPLAY#
```

The command will not produce any output. Confirm the license has returned by displaying the license status again in the **Ansys Client Licensing Settings** application.