Instructions on how to select an ANSYS License

ANSYS provides an explicit finite element solver for structural analysis (ANSYS FEA) and a fluid dynamic solver (ANSYS FLUENT) for solving compressible and incompressible fluid flows. While both products are packaged under the ANSYS v12 software label, they have a complicated license structure that needs to be understood before using the product.

The College of Engineering has the following licenses:

- ANSYS Academic Research Mechanical and CFD (Quantity = 50)
- ANSYS Academic Teaching Mechanical and CFD (Quantity = 500)
- ANSYS Academic Meshing Tools (Quantity = 5)
- ANSYS HPC (for parallel processing) (Quantity = 256)

Any course based work should use ANSYS Academic Teaching licenses. Any 999 or general research should use the Academic Research license.

Instructors must inform their class on which license they should use. Lab operators do not know which licenses should be used by which classes.

To select the license, users should follow the steps outlined below:
Step 1) From the START menu, select

“ANSYS 15.0” -> “ANSYS License Preferences” -> “User License Preferences”

Step 2) When the Client ANSLIC_ADMIN Utility starts, select “Set License Preferences for User name.”
Step 3) When the “License Preferences” appears, select “ANSYS Academic Teaching Advanced” and click the “Move up” button until it is at the top of the list.

Step 4) Do the same for “ANSYS Academic Research CFD” and “ANSYS Academic Research” until your screen looks similar to the one below:

![License Preferences Window](image)

Step 5) Click “Apply” and then “OK”

Step 6) Start the ANSYS solver

NOTE: If you get an error message that your model is too big, you will need to set “ANSYS Academic Research CFD” and “ANSYS Academic Research” as #1 and #2 in the list.