<u>Common Commands in ANSYS APDL</u>

- Selecting Stuff – these commands can be used in any processing section
  - ***COMMAND*,(S or A)** – for any selection command below, the first keyoption will always be S or A.   S = select a new set (default)   A = Additionally select a set and extend the current set.
  - **ALLSEL** – Selects all entities with a single command (<u>*always do this after performing an action on a set of selected entities*</u>)
  - **NSEL**– Selects a subset of nodes . Example: if selecting all nodes at a location such as in the XZ plane – NSEL,S,LOC,Y,0 *check help topics for options e.g. node at kp
  - **ASEL** -- Selects a subset of areas. Example: if you have 1 area selected and then plotted areas and their numbers and want to select an additional area by its number – ASEL,A,AREA,#
  - **LSEL** – Same as ASEL, but for lines.
  - **KSEL** – Same as ASEL, but for keypoints.
  - **ESLL** – Selects those elements associated with the selected lines.
  - **ESLN** – Selects those elements attached to the selected nodes.
  - **NSLE** – Selects those nodes attached to the selected elements.
- Plot settings
  - **/PNUM**,(label),(on/off) – Controls entity numbering/coloring on plots.
    - label – NODE, ELEM, KP, LINE, AREA
    - on/off – 1 = on, 0 = off
  - To turn the background from black to white in order to capture images and print them without wasting ink, copy the following lines into the beginning of every script file as the line below /CLEAR ->
    - /RGB,INDEX,100,100,100, 0
    - /RGB,INDEX, 80, 80, 80,13
    - /RGB,INDEX, 60, 60, 60,14
    - /RGB,INDEX, 0, 0, 0,15
  - **/ESHAPE,1** – Change plot settings to display element shapes using section data.
  - **/VIEW,1,1,1,1** – Change view settings to isometric.
  - **/REPLOT** – Replot the current plot screen. Use this command after changing any plot settings to see an updated plot with current settings.
- To plot items:
  - **EPLOT** – Displays the selected elements (if ALLSEL the line before, this command will display all the elements in the system.
  - **KPLOT** – Displays the selected keypoints.
  - **LPLOT** – Displays the selected lines.
  - **APLOT** – Displays the selected areas.
- General commands**:**
  - **!** – Comments out script after it on the same line after a command. Type a space before it.
  - **FINISH** – Exits normally from a processor. Always put as 1st line in the script file.
  - **/CLEAR** – Clears the database. Always put as the 2nd line in the script file.
- **/PREP7** – Enters the model creation preprocessor.
- Physical properties of system:
  - **ET** – Defines a local element type from the element library.
  - **SECTYPE** – pick from predefined list for cross section, e.g. rectangular beam.
  - **SECDATA** – specify relevant data, e.g. dimension of beam cross-section.
  - **MP,**(label) – Defines a linear material property as a constant or a function of temperature. Label e.g. EX or PRXY.
- Modeling (Geometric/Physical Model):
  - **K**,#1,#2,#3,#4 – Defines a keypoint. #1=NumericalLabelForKeypoint, #2=X-Coordinate, #3=Y-Coordinate, #4=Z-Coordinate
  - **L**,#1,#2 – Create line between keypoints. #1 = 1st Keypoint   #2 = 2nd Keypoint
  - **A** – Defines an area by connecting keypoints
    - A,(kp#1),(kp#2),…(up to 18) – list kp in counter-clockwise order because orientation of area plane follows right-hand rule
  - **AL** – Generates an area bounded by previously defined lines. Typically used by selecting all lines via LSEL then create area by: AL,ALL

        - **CIRCLE** – Generates circular arc lines
        - **BLC4** – Generates a rectangular area or block volume by corner points
        - **LFILLT** – Generates a fillet line between two intersecting lines Template: LFILLT,(line#1),(line#2),(FilletRadius)
    - Meshing (Finite Element Model):
        - **N** – Defines a node. Template: N,,(x-coordinate),(y-coordinate),(z-coordinate)
        - **E**,(node#1),(node#2) – Defines an element by node connectivity.
        - **TYPE**,# – Sets the current element type to mesh with. # indicates the ET as previously defined.
        - **REAL**,1 – Sets the current element real type to 1 (this is the default so you don't typically need to include this command in the code before meshing).
        - **MAT**,# – Sets the current material properties according to the material # as previously defined.
        - **SECN**,# -- Sets the current section type and section data as previously defined. For example, use this if modeling a wooden truss with different types of beams, even though they are all the same element.
        - **ESIZE** – (Element Size) Specifies the default number of line divisions.
        - **LMESH** – (Line Mesh) Generates nodes and line elements along lines
        - **AMESH** – (Area Mesh) Generates nodes and area elements within areas
        - **SMRTSIZE**,# – Specifies meshing parameters for automatic (smart) element sizing. #: 1-10   1=fine, 10=coarse
        - Refining mesh using smart meshing:
            - **LREFINE**,#1,#2 – Refines the mesh around specific lines. Typically, select lines first by using LSEL. Then, #1=all, #2=1-10 level of smart mesh to refine to.
            - **AREFINE**,#,# – Same as LREFINE, but for areas.
            - **KREFINE**,#,# – Same as LREFINE, but for keypoints.
    - Boundary conditions and loads:
        - **DK** – Defines degree-of-freedom constraints at keypoints. Template: DK,(kp# or ALL if selected set already using KSEL),(UX, UY, UZ, ROTX, ROTY, ROTZ),(value which is usually zero because this helps is define boundary conditions). See help topics for all available degree-of-freedom constraint options
        - **FK** – Defines force loads at keypoints. Template: FK,(kp# or ALL if selected set already using KSEL),(FX, FY, FZ, MX, MY, MZ),(value of force). See help topics for all available loading options
        - **D** – Defines degree-of-freedom constraints at nodes.
        - **F** – Same as FK, but for nodes.
        - **SF** – Specifies surface loads on nodes (e.g., used for distributed loads).
- **/SOLU**
    - **SOLVE** – Starts a solution. If solution ran successfully, there will be a little pop-up window that says "Solution is done!"
- **/POST1**
    - Plot results
        - **PLESOL** – Displays solution results as discontinuous element contours. Help topics for results options
        - **PLNSOL** – Displays solution results as continuous element contours. "Help Topics" for results options
    - Print Results as text
        - **PRNSOL** – Prints nodal solution results. See help topics for all available results
        - **PRESOL** – Prints the solution results for elements. *See help topics for all available results*
    - Create paths and display path data:
        - **PATH** – Defines a path name and establishes parameters for the path. Template: PATH,(name for path max. 8 characters),(# of points to define path, default is 2),30,(# of divisions between adjacent points, default 20)
        - **PPATH** – Defines path. Template: PPATH,(point # 1 or 2),(node # but leave blank if using xyz loc),(x-coordinate),(y-coordinate),(z-coordinate)
        - **PDEF** – Interpolates a path item onto a path. Example of saving data values of VonMises stress along a path to a certain name that can be referenced later: PDEF,VONSTRSS,S,EQV,AVG
        - **PLPATH** – Displays path items on a graph. Template: PLPATH,S,(predefined path data name from PDEF)
        - **PRPATH** – Prints path items along a geometry path. Template: PRPATH,S,(predefined path data name from PDEF)