Gerald Yoho
ASEN 3802
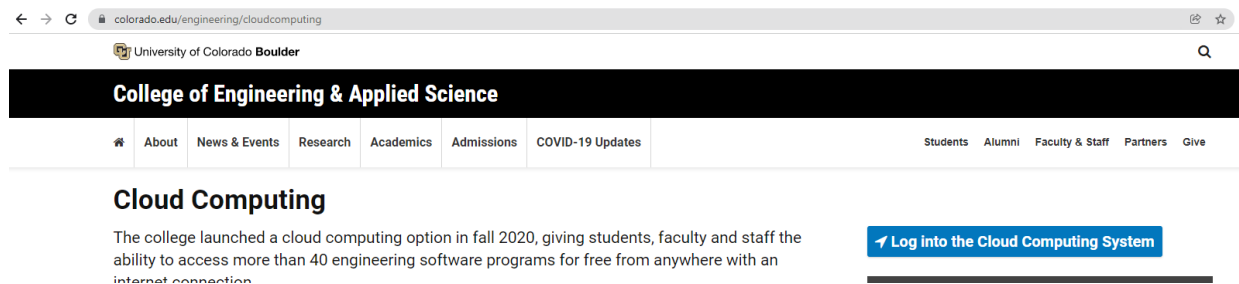Lab 2: ANSYS Tutorial Revision
8/30/2023

1. <mark>Most Important: **SAVE OFTEN!!!**</mark>
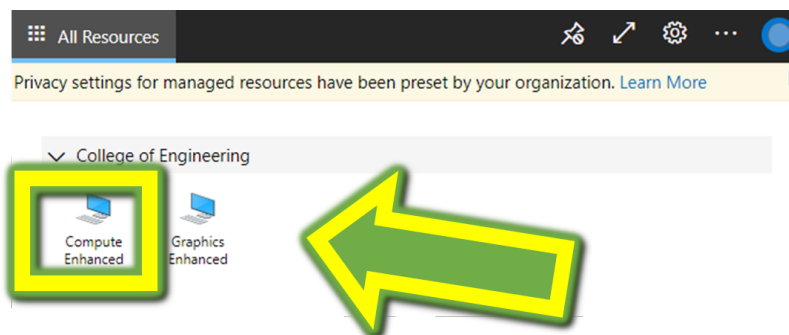

2. Open **ANSYS Mechanical APDL 2022 R2**
   - Use free student version
     https://www.ansys.com/academic/free-student-products
   - Or use CU Boulder Cloud Computing
     Start Menu > ANSYS 2022 R2 > Mechanical APDL 2022 R2

Accessing Software Remotely
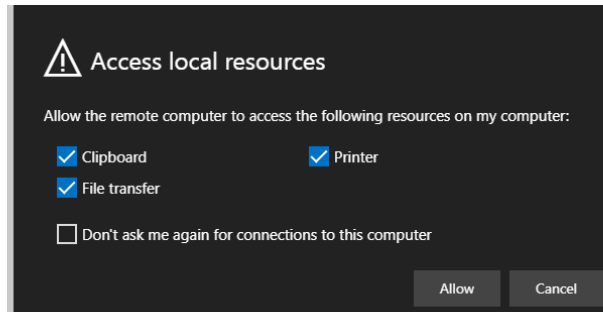
https://www.colorado.edu/engineering/cloudcomputing

- Click the *"Log into the Cloud Computing System"*
  - ***When Prompted, enter your Identikey credentials***

- ***Login will ask you to select "Compute Enhanced" or "Graphics Enhanced"***
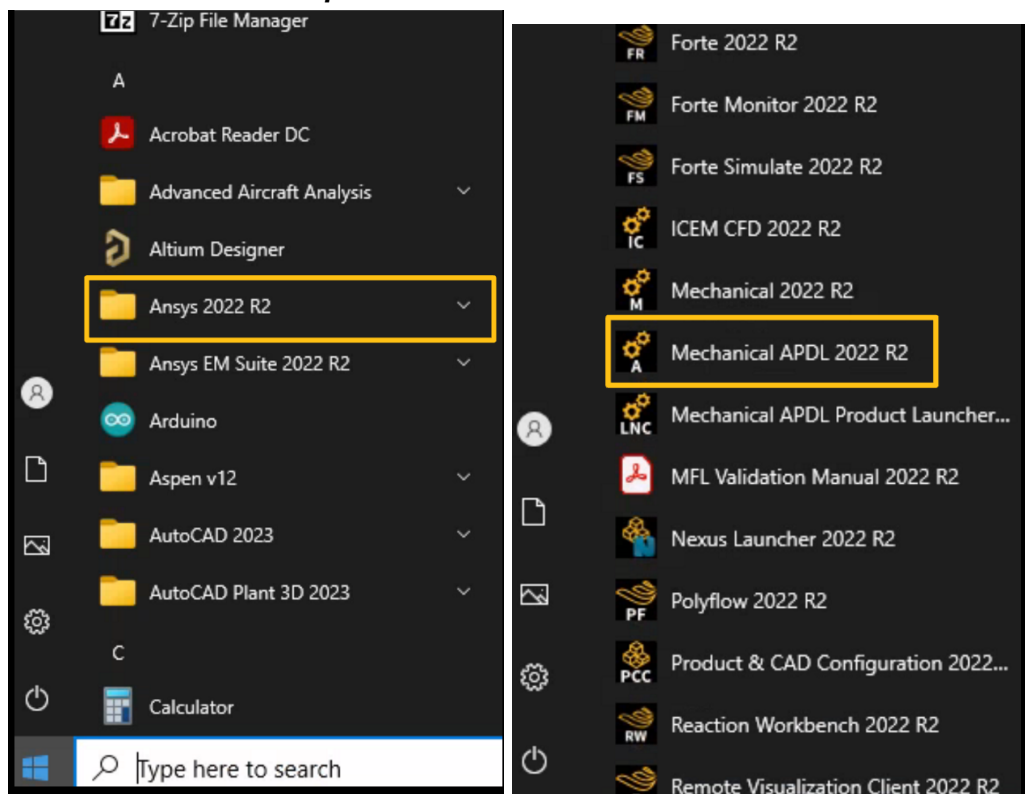 - Select ***"Compute Enhanced"*** *(Both Compute Enhanced and Graphics Enhanced will work)*

● *Allow access to local resources*

● *Enter you Identikey Credentials again*

Operates as a normal PC

Click Windows Icon > Open ANSYS Folder > ANSYS Mechanical APDL 2022 R2

Save your DATA/FILES to the cloud before you log out!

once you log off - all your data/files will be deleted from this remote desktop

This tutorial is intended to help familiarize you with the user interface for this ANSYS package but is not a full step-by-step guide for your lab.

3. <u>Get familiar with the "Preprocessor" tab on the left-hand side of the screen</u>
   - a lot of this tutorial (and your lab) will take place in this area
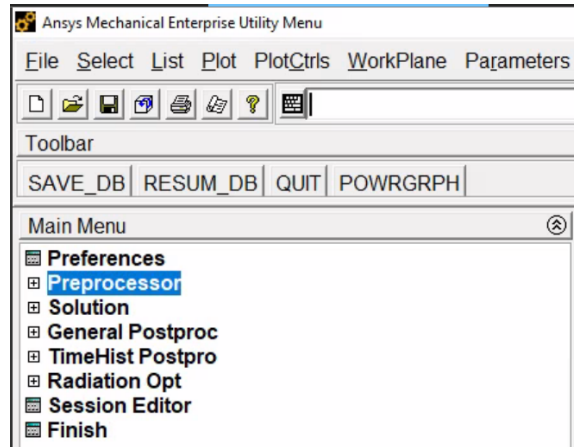
4. If at any point your model doesn't automatically refresh or load after updating nodes, elements, etc. you can use the ANSYS top menu **Plot > Replot** to manually force an update

5. <u>Setup & Initializing</u>
   a. In the ANSYS <u>Main Menu Tree</u>:
      - **Preprocessor > Element Type > Add/Edit/Delete**
      - In the pop-up window click **Add**
      - Select the 3D finit stn 180 element under the Structural Mass/Link category.
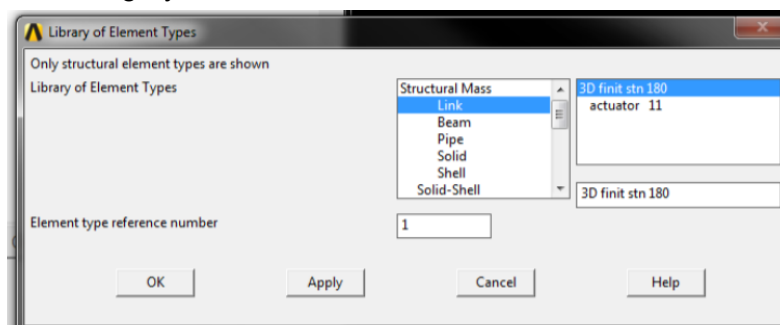
      - Click **Apply, Cancel, & Close**

b. Command Window

- In the command window type in the following: "**SECTYPE, 0, LINK, ,
  LINK1,** "
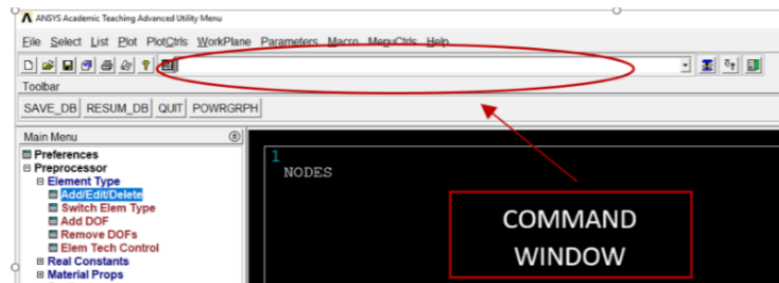   - All of these commas are necessary; keep them
    - Do not include the quotation marks
    - Use capital letters
    - Note: SECTYPE sets the section ID number, section type,
      subtype for a section (nothing in this case) and section name.
- Hit enter
- In the command window type in the following: "**SECDATA, 2.5**"
    - Note: SECDATA defines a constant cross section area for each
      spar element (In this case set to 2.5 Units). You do not
      specify a unit system. Instead, ANSYS assumes a consistent
      unit set. Thus, for metric problems, enter all values in
      [m]/[kg]/[s]. (i.e. area in [m2], to produce displacements with in [m])
- Hit enter

c. Define Material Properties (in the ANSYS <u>Main Menu Tree</u> again):
- **Preprocessor > Material Props > Material Models**
    - A window pops up

- In this new window, select **Structural > Linear > Elastic > Isotropic**
    - A second window pops up

- In the second pop-up window enter the Young's Modulus into the **EX** field **1.3E7**.

- Leave the **PRXY** (Poisson's Ratio) field **blank**, thereby assuming a value of 0
    - Select OK
   - Cose out of the remaining "Define Material Modal Behavior" Window
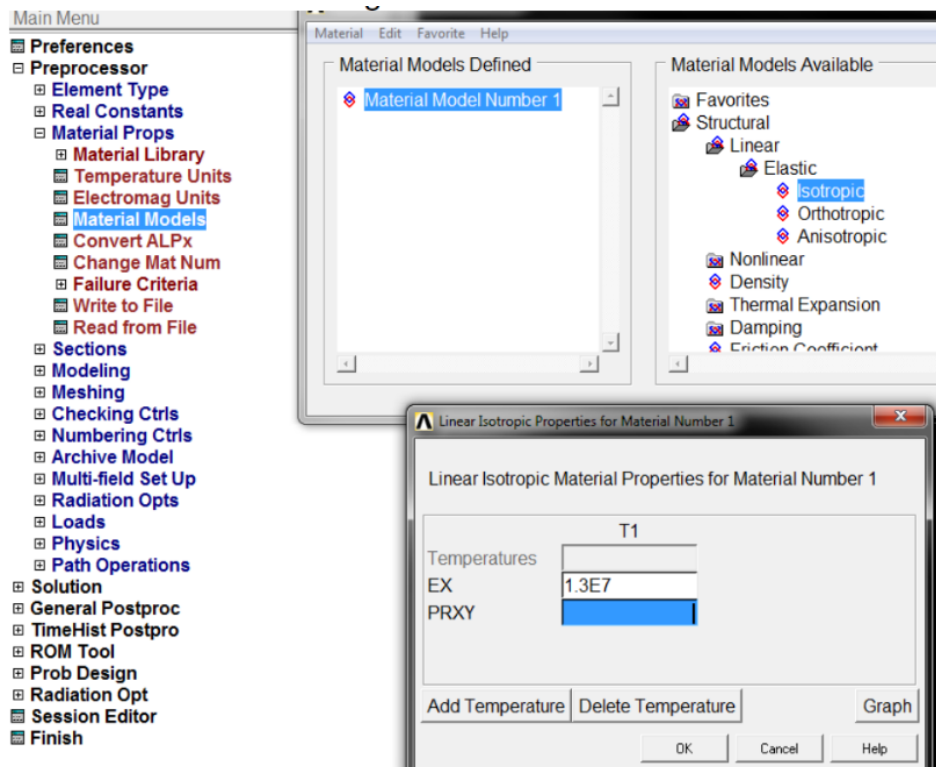
6. Let's add some elements to the schematic!

  We'll be modeling a 2-D, 4 cell truss. *Most of this happens in the **Preprocessor** tab*

   a. Nodes

   - **Preprocessor > Modeling > Create > Nodes > In Active CS**
   - Enter the first node number (1)
     - Enter the coordinates of node 1: x = 0, y= 0
     - Click **Apply**
     - Create nodes 5, 6 & 10
       - Node 5: x = 80 y = 0
       - Node 6: x = 0 y = 20
       - Node 10: x = 80 y = 20

Ansys script (an input file of commands):
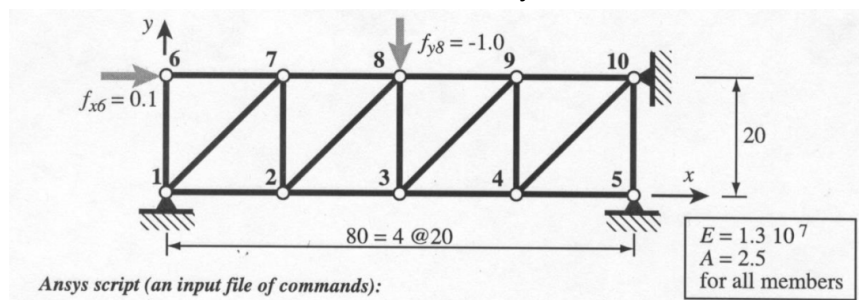
   - Now we'll auto-fill the nodes between 1 and 5 as well as nodes 6-10
   - **Preprocessor > Modeling > Create > Nodes > Fill between Nds**
   - Make sure "**pick**" is selected
     - In the main window, click on nodes **1** & **5** as the nodes we will auto populate > Click OK
     - Input "**2**" for **NSTRT**
       - This will start at node 2 and finish at node 4
       - Click **Apply**
     - Repeat by selecting nodes **6** & **10**
     - Input "**7**" for **NSTRT**
       - This will start at node 7 and finish at node 9
       - Click **Apply**

DON'T FORGET TO SAVE!

b. Lines

- **Preprocessor > Modeling > Create > Elements > Auto Numbered >Thru Nodes**
- Make sure "**pick**" is selected
- Click nodes **1** & **2**
  - Click **Apply**
  - This just makes a line connecting the two selected nodes
- Repeat for nodes **1** & **6**, **1** & **7**, **6** & **7** and **5** & **10**
- At this point, the design in your center window should look similar to the image to the right (with an additional line between nodes 5 and 10).
- Now we'll copy the shape we've just created
- **Preprocessor > Modeling > Copy > Elements > Auto Numbered**
- Select the 4 elements shown in the figure above
  - (lines between **1** & **2**, **1** & **6**, **1** & **7**, **6** & **7** but <u>not</u> between 5 & 10)
- Click **OK**
- Enter "**4**" for **ITIME**
  - Creates 4 "unit cells" (three copies plus the one we already made)
- Enter "**1**" for **NINC**
  - Increments node number by one for each copy
- Click **OK**

- If we wanted to define different material properties for different sets of elements, we could do this by using:
  - **Preprocessor > Modeling > Create > Elements > Element Attributes**
  - We won't be using this at this point but if you wanted to this is where you'd do it

DON'T
FORGET TO
SAVE!
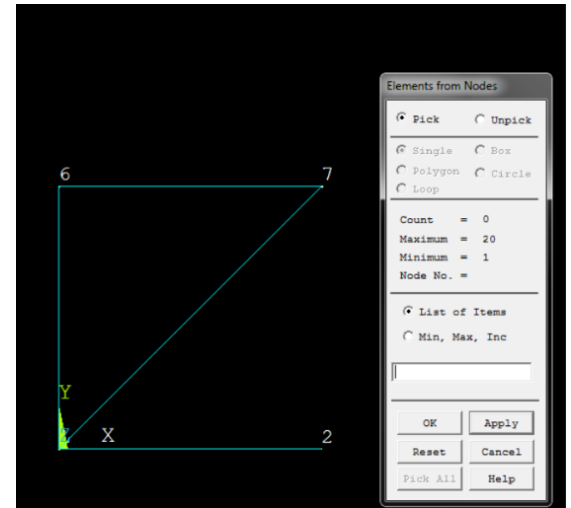
7. <u>Defining Boundary Conditions & External Loads</u>
   Instead of "preprocessor", we'll move our attention to a new tab: "Solution"

   a. Boundary Conditions
      ● **Solution > Define Loads > Apply > Structural > Displacement > On Nodes**
      ● Select nodes **6, 5,** & **10**
      ● Click **OK**
      ● Select **ALL DOF, Constant Value,** and enter **0**.
        ○ This applies a fixed boundary condition to joints 1, 5, and 10
      ● Click **OK**

   b. External Forces/Moments

      *This version of ANSYS has an issue/bug when trying to execute "normally". Not to worry, I've provided you with a work around that actually makes it easier (in my opinion at least)

      i. The version that works: **command line window**
      ● Type "**F, 6, FX, 0.1**" and hit **Enter**
        ○ Include the commas, not the quotation marks
        ○ This specifies a force at node 6 in the X direction (rightward) with a magnitude of 0.1
      ● Type "**F, 8, FY, -1**" and hit **Enter**
        ○ Include the commas, not the quotation marks
        ○ This specifies a force at node 8 in the negative Y direction (downward) with a magnitude of 1
      ● That's it! Super easy, right?

      ii. The "normal" way that is **<u>broken</u>** with this version of ANSYS (you can skip this but it's good to know how to use it)
         ● **Solution > Define Loads > Apply > Structural > Force/Moment > On Nodes**
         ● Select Node **6** and click **OK**
            ○ Select **FX** and **Constant value**
            ○ Enter **0.1** as the magnitude
            ○ Click **Apply**
         ● Select Node **8** and click **OK**
            ○ Select **FY** and **Constant value**
            ○ Enter **-1** as the magnitude
            ○ Click **Apply**
         ● Click **OK**

8. <u>Solve the Steady State Structural Finite Element Problem</u>
- **Solution > Solve > Current LS**
  - Solves the specified loading conditions
  - Click **OK**

***Should you get an error message at this or any point, check your model. You can use the items under **List** in the <u>top </u>menu to obtain the nodes, elements, and loads. If your model is incorrect add missing items and/or remove items; see delete option under the menus that you have used to create nodes, elements, and loads.*

9. <u>List and Visualize Results</u> *(The rest of this document is just plots, results, etc)*
  - *Instead of "preprocessor", we'll move our attention to a new tab: "General Postproc"*
- Expand **General Postproc > List Results > Nodal Solution**
- Select **DOF Solution > Displacement Vector Sum**, Click **OK**
  - This tabulates the x and y direction displacement values for each node in the model
  - If you want to save these results, you can create a screenshot using the Windows snipping tool, or save the image

- Expand **General Postproc > List Results > Reaction Solu > All struc forc**
- Click **OK**
  - This tabulates the x and y reaction forces for each supported node in the model; make sure that the forces reported are in balance with the applied forces.
  - If you want to save these results, you can create a screenshot using the Windows snipping tool, or save the image

- Expand **General Postproc > List Results > Nodal Loads > All struc forc F**
- Click **OK**
  - This tabulates the nodal forces for each node in the model; make sure that the forces reported for the loaded nodes (6, 8) show the correct values.
  - If you want to save these results, you can create a screenshot using the Windows snipping tool, or save the image
  - *I was getting an error for this*
    - Try in the command window: "**SET, LAST**"

- Expand **General Postproc > ElementalTable > Define Table > Add**
  - In the **Item, Comp, Results table**, choose **By Sequence Number** and select **SMISC**. After **SMISC**, input **1**. (So it will look like "**SMISC, 1**" The meaning of "1" is defined in the ANSYS user manual that can be accessed via the "Help" menu.) Then **Apply, Close**, etc.
  - This gives the element forces

- ○ *I was getting errors for this*
    - ■ Try in the command window: "**SET, LAST**"
    - ■ https://forum.ansys.com/discussion/13263/the-requested-database-is-not-available-the-etable-command-is-ignored
- ○ Similarly, we could solve for stress

- ● Expand **General Postproc > Plot Results > Deformed Shape**
- ● Click **Def + Undeformed**, then click **OK**
    - ○ A visual representation of the deformed shape and a comparison to the original shape allows you to quickly check the results against your intuition, sometimes catching minor mistakes in the FEM model.

- ● Expand **General Postproc > Plot Results > Contour Plot > ElemTable**
- ● Click **OK**
    - ○ A visual representation of the forces in the bars will be shown
    - ○ *I'm getting an error when running this*
        - ■ Try in the command window: "**SET, LAST**"

- ● You can get the values of the bar forces from:
    - ○ **General Postproc > List Results > Elem Table Data > SMIS1**