ASEN 3112 Spring 2020 – ANSYS Tutorial for Lab #2

Objective:

      This tutorial aims to familiarize students with basic ANSYS functions by modeling a 2-D pinned truss described by figure 1. While similar to the 16-bay space truss, key differences in the modeling procedure exist. Thus, this tutorial also highlights areas where the student should consider the possible changes required to accurately model the space truss.
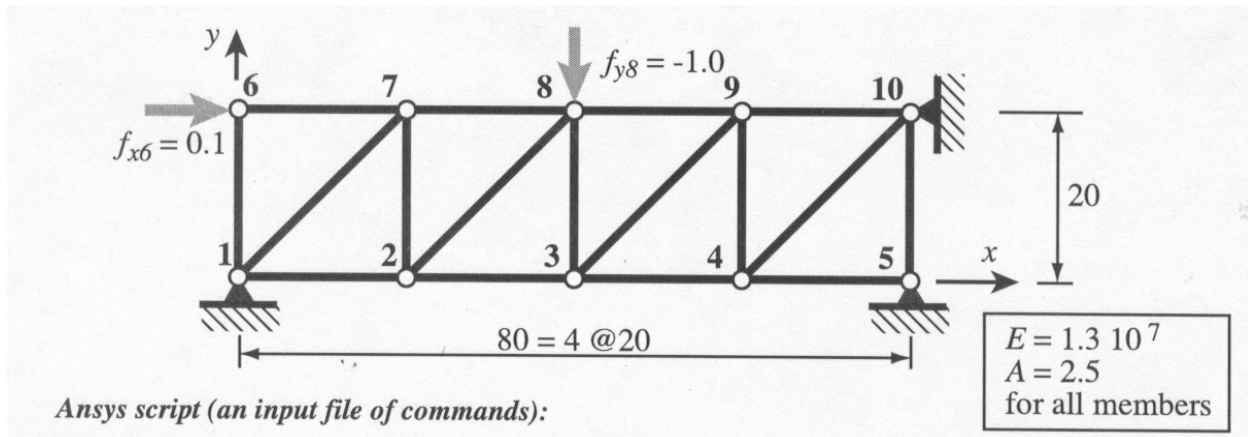

Ansys script (an input file of commands):

Figure 1: Problem Definition & Geometry

Step 1: Open ANSYS Mechanical APDL

- From Desktop:
- Click Microsoft Windows Symbol on the bottom left corner of the screen
- Click on "ANSYS 19.1" Folder
- Scroll down until you find the "Mechanical APDL 19.1" Application
- Click on: "Mechanical APDL 19.1"
- Click Okay on any subsequent user prompts until the following ANSYS Main Menu Screen is reached (Figure 2)

Step 2: Setup Files & Familiarize Yourself with ANSYS Mechanical APDL

- Take a few minutes to explore the working environment. The ANSYS Main Menu, along the left side, provides a hierarchical command tree. First, define the problem in the **Preprocessor**, then apply the boundary conditions and solve the load situation in the **Solver**, and finally review the results under **General Postprocessor.** The central window provides visual output. The toolbar at the top accesses utilities. **File** allows you to save, load, and exit. **List**, provides problem data tables, such as nodal coordinates. Plot outputs data to the central window.
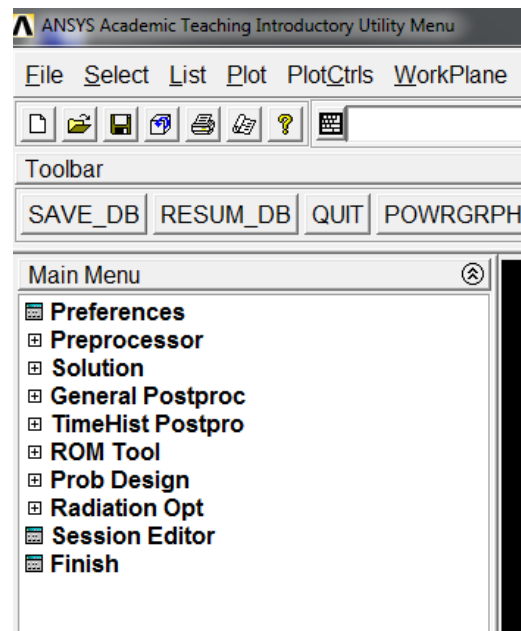

Figure 2: ANSYS Main Menu

Step 3: Select Element Type, Set Element Constants & Material Properties

- Under the **ANSYS Main Menu** tree:
- Expand the menu tree by clicking **(+)Preprocessor**, and then **(+)Element Type** in the second tree level. Select the **Add/Edit/Delete** command, as illustrated in Figure 3.
- In the pop-up window click **Add**
- Select the **3D finit stn 180** element under the **Structural Mass/Link** category.
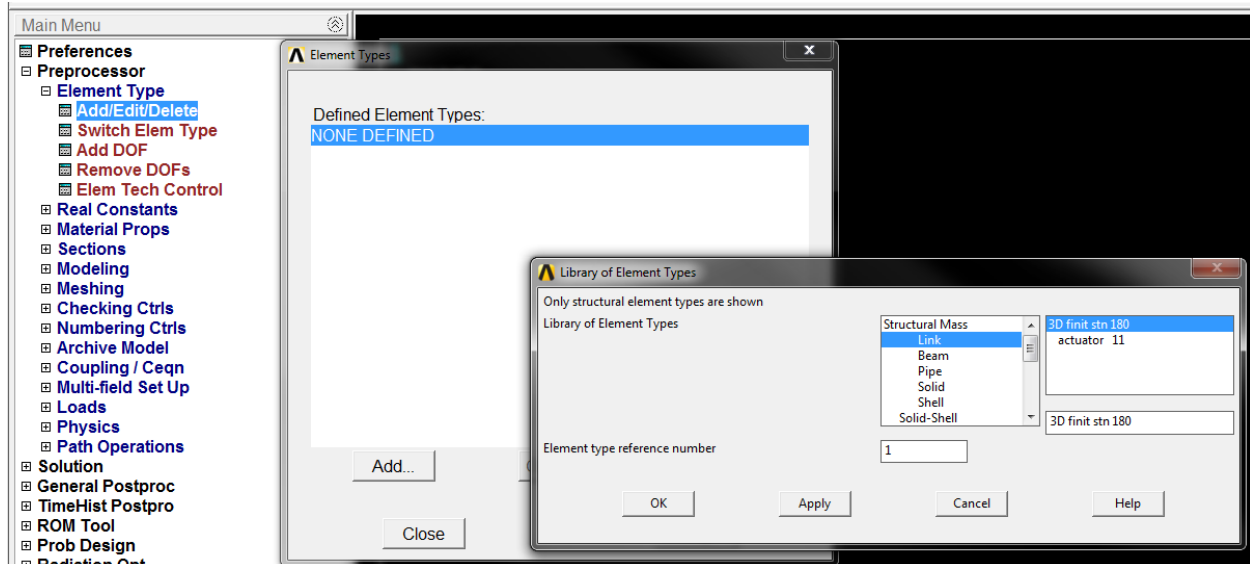
Figure 3: Element Type

- Click **Apply, Cancel, & Close**
- In the command window type in the following: "SECTYPE, 0, LINK, , LINK1, " (Yes, the placement of these commas are correct)
- Hit enter
- Note* **SECTYPE** sets the section ID number, section type, and subtype for a section.
- In the command window type in the following: "SECDATA, 2.5"
- Hit enter
- 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 units 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]).
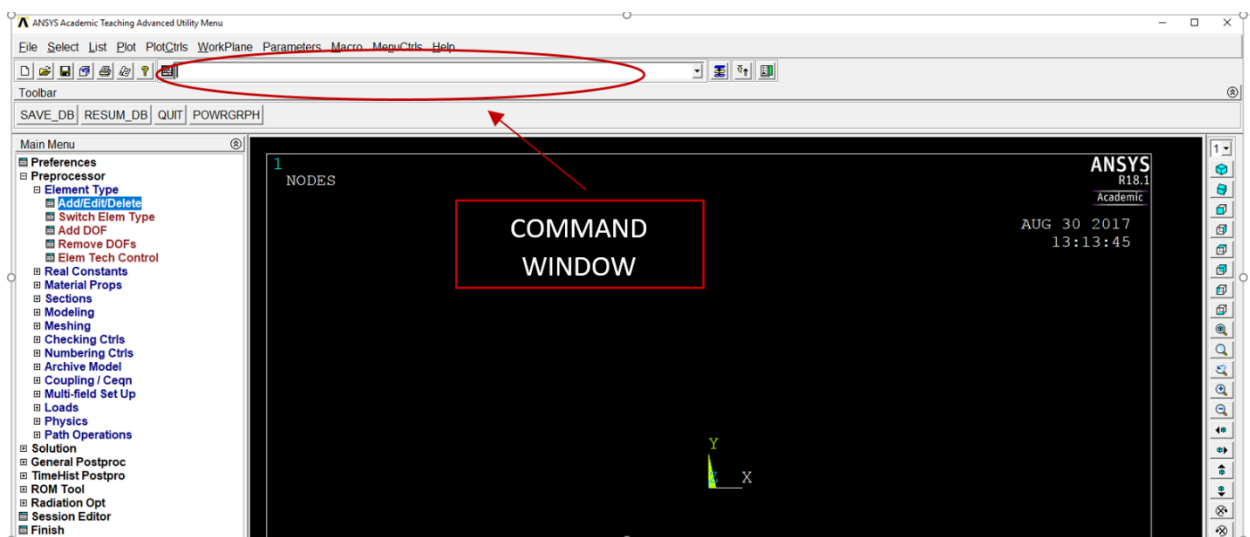

Figure 4: Command Window

- Click **(+)Material Props** in the second level under **(+)Preprocessor**, and then execute the **Material Models** command by clicking on it.
- Select, in order, **Structural/Linear/Elastic/Isotropic** in the pop-up window, as shown in Figure 5. This defines the material behavior model used for each element it is assigned to. For this simulation, the material will exhibit a perfect linear elastic stress/strain relationship, with elastic constants uniform in each direction.
- 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**
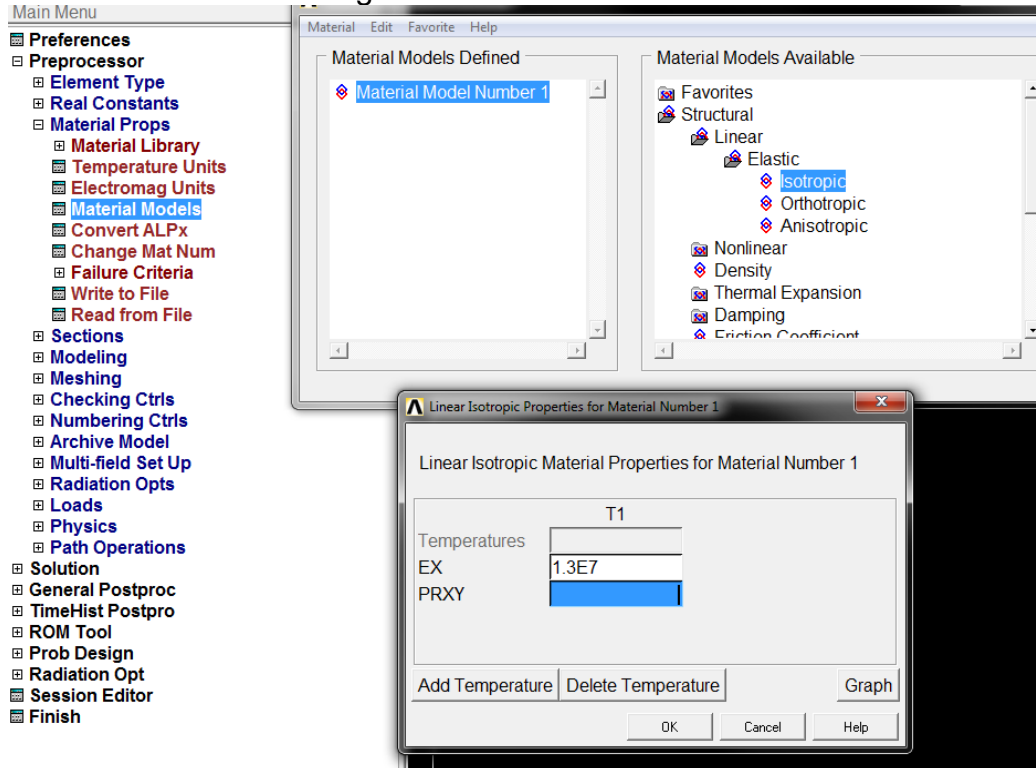- Close out of the remaining "Define Material Modal Behavior" Window

Figure 5: Material Properties

Reminder: Please save your work often by selecting **Save as Jobname.db** under **File**

Step 4: Define the Problem Geometry

- With the **(+)Preprocessor** level still open, expand **(+)Modeling**, then **(+)Create**, and finally **(+)Nodes**. Click the **In Active CS** (coordinate system) command
- Enter **1** for the node number
- Enter x=**0**, y=**0** in the first two fields
- Click **Apply.** This creates a node at the origin, corresponding to joint 1 in Figure 1.
- Repeat the procedure to create nodes 5, 6, & 10 to represent joints 5, 6, & 10 (use Figure 1)
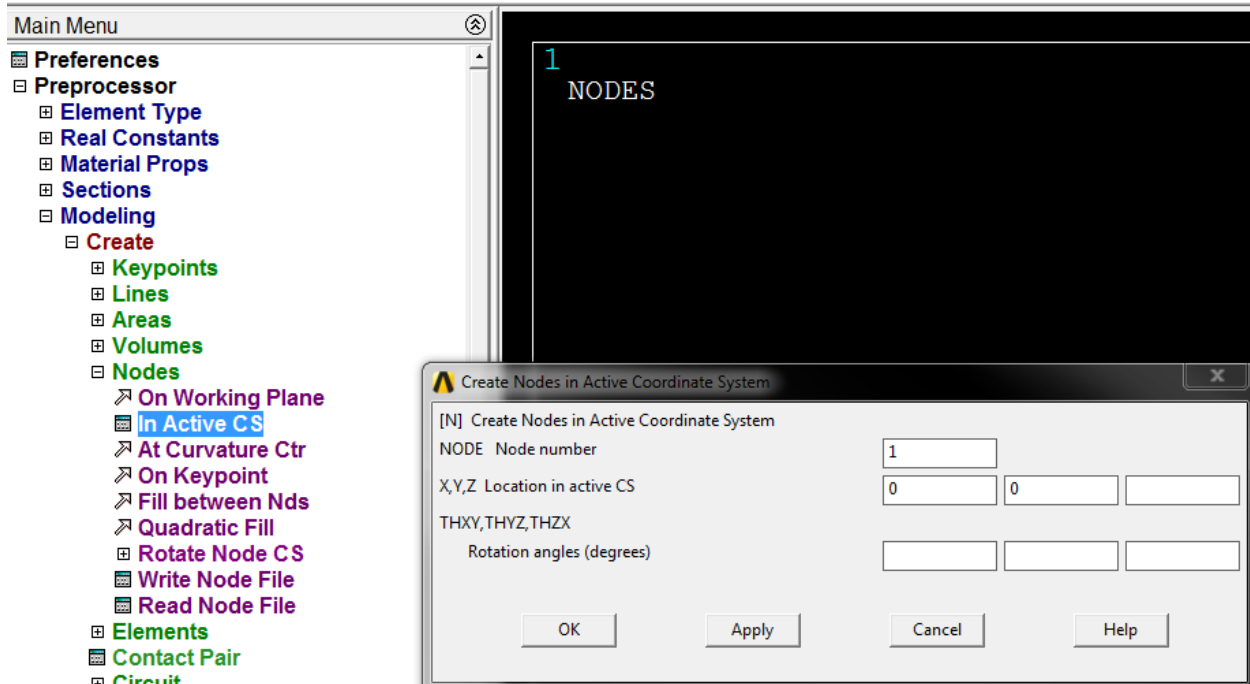
Figure 6: Creation of Nodes

- Notice each node plotted in the central visual output window
- Confirm the nodal coordinates by using the menu item under **List**
- With the tree expanded, as in the previous step, select the **Fill between Nds** option
- Verify **Pick** is selected and then click on nodes 6 & 10 in the center window
- Click **OK**
- Enter 3 for **NFILL** to create three nodes between 6 & 10
- Specify 7 for **NSTRT** to start at node 7 and finish with node 9
- Enter 1 for **SPACE** to provide uniform spacing
- Click **Apply**
- Repeat procedure with nodes 1 & 5
- This time specify 2 for **NSTRT** to start at node 2 and finish with node 4

Note: You should see at this point the nodes with node IDs on the display. If this is not the case, try:

1. Top horizontal menu: "Plot", then "Replot"
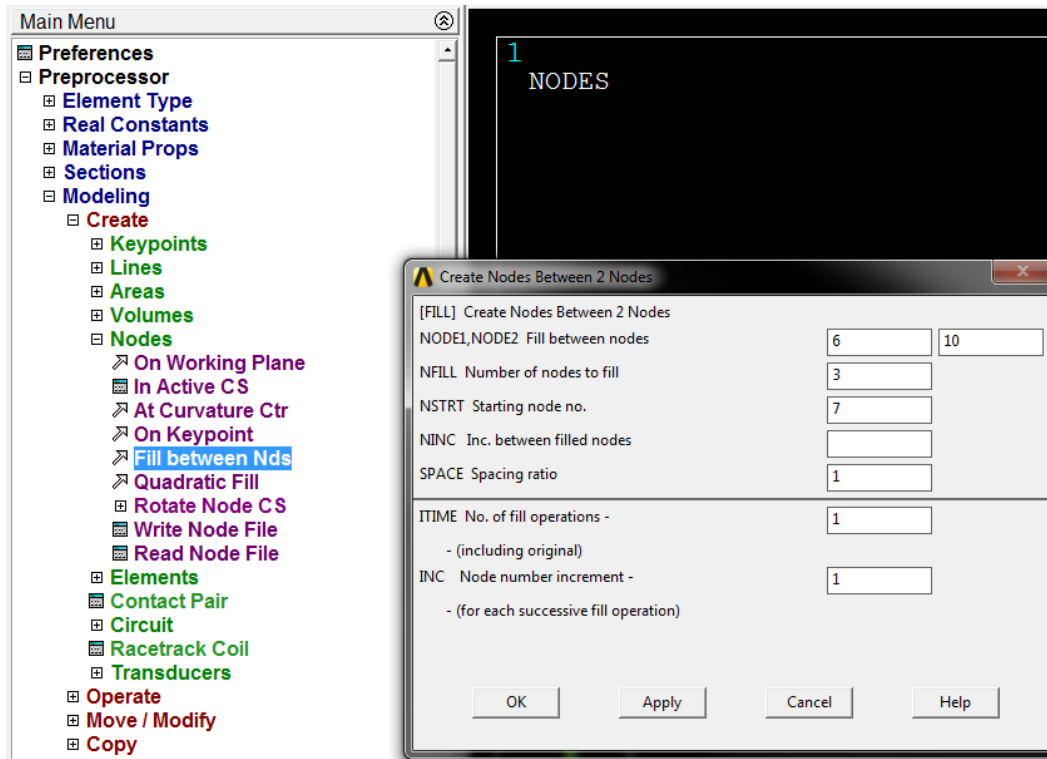2. Top horizontal menu: "PlotCtrls", then "Pan Zoom Rotate", then "Fit" (lower left corner of window)

Figure 7: Fill Between Corner Nodes

- Define a unit cell, the smallest repeating pattern in the truss assembly
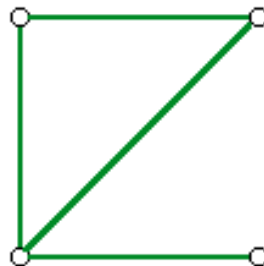- For this case, Figure 8 displays the unit cell

Figure 8: Unit Cell

Question: What is the unit cell for the 16-bay space truss? Sketch the unit cell in the box bellow

- Expand **(+)Elements** in the fourth level under the already expanded **(+)Preprocessor/ (+)Modeling/(+)Create** path, and then click **(+)Auto Numbered** in the fifth level. Execute the **Thru Nodes** command
- Select **Pick** and then click on nodes 1 & 2. Click **Apply**. This creates an element with end nodes 1 and 2
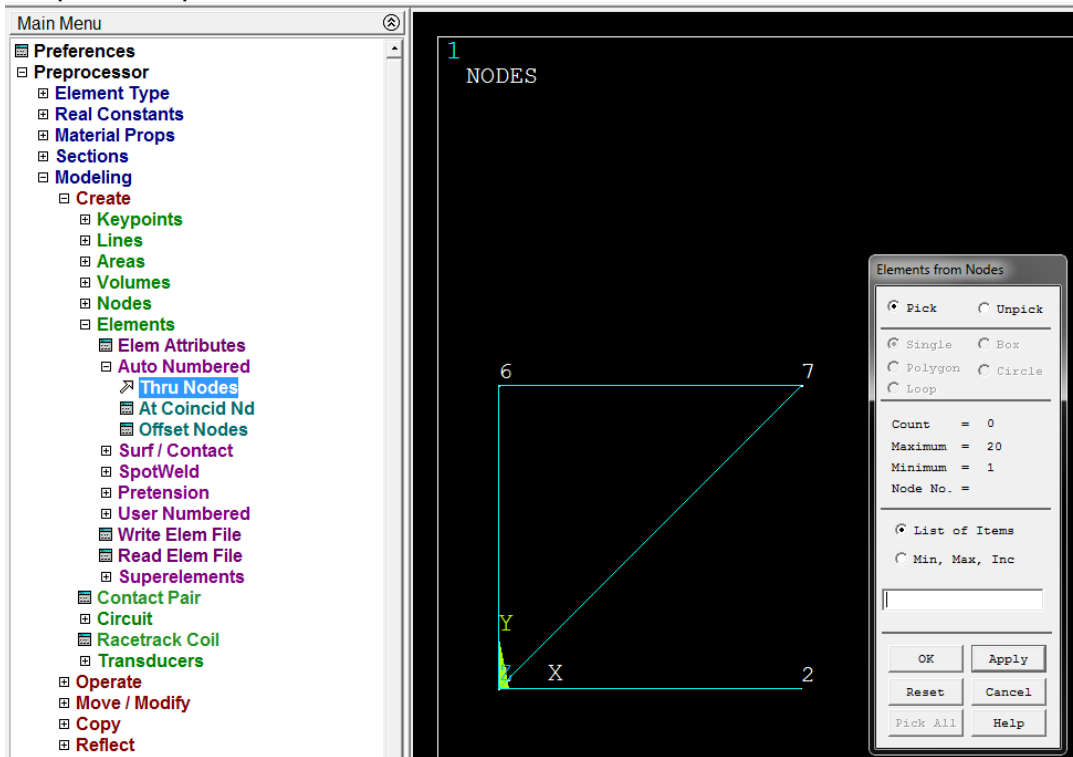- Repeat the procedure with nodes 1 & 6, 1 & 7, and 6 & 7

Figure 9: Creation of Unit Cell

- Click on **(+)Copy** in the third level under **(+)Preprocessor/(+)Modeling**, and then **(+)Elements** in the next level. Click the **Auto Numbered** command
- Select each of the four existing elements by clicking on each of them
- Click **OK**
- Enter 4 for **ITIME,** to create four unit cells (three additional copies plus the original)
- Specify 1 for NINC, to increment the node number by one for each copy (i.e. the position occupied by node 7 in the original cell is occupied by node 8 in the first copy).
- Click **OK**
- Create the last element through node 5 & 10 to complete the truss

Note: You should see at this point the nodes and the bars on the display. If this is not the case, try:

1. Top horizontal menu: "Plot", then "Replot"
2. Top horizontal menu: "PlotCtrls", then "Pan Zoom Rotate", then "Fit" (lower left corner of window)
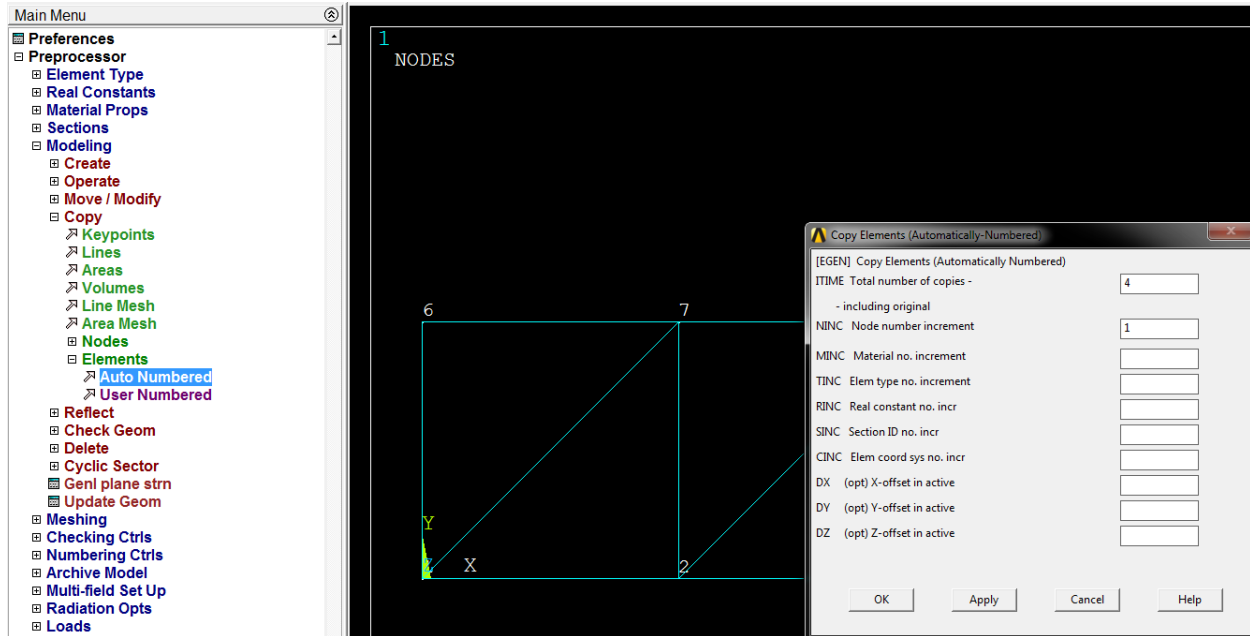
Figure 10: Copying Unit Cell

- Under, the still expanded **(+)Preprocessor/(+)Modeling/(+)Create/(+)Elements** path, select **Element Attribute**
- This provides a way to specify different material properties and element constants for each set of elements in the model (i.e. modeling a structure composed of multiple materials)
- Cancel

Step 5: Define the Boundary Conditions & External Loads

- Click in successive levels **(+)Solution**, **(+)Define Loads**, **(+)Apply**, **(+)Structural**, and **(+)Displacement**. Click **On Nodes**
- Select nodes 1, 5, and 10
- Click **OK**
- Select **ALL DOF, Constant Value**, and enter 0. This applies a fixed boundary condition to joints 1, 5, and 10 as specified by Figure 1.
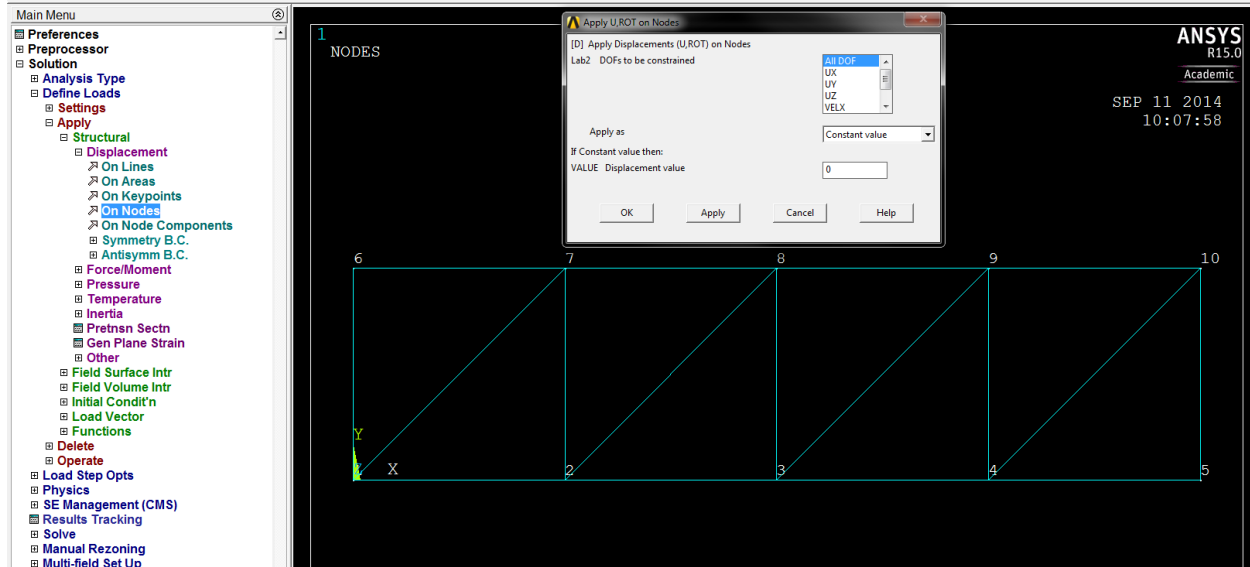- **OK**

Figure 11: Apply Displacement Boundary Conditions

- Click **(+)Force/Moment** under the expanded **(+)Solution/ (+)Define Loads/ (+)Apply/(+)Structural** path. Execute the **On Nodes** command
- Select node 6
- Click **OK**
- Select **FX** to apply a x-direction external force, constant value
- Enter 0.1, the magnitude of the force vector, in the **VALUE** field
- Click **Apply**
- Repeat the process for node 8, use Figure 1 for correct load configuration
- Select **FY**
- Enter -1 in **VALUE** field
- Click **Apply**
- Click **Ok**

Step 6: Solve the Steady State Structural Finite Element Problem

- With the **(+)Solution** level still expanded, open **(+)Solve.** Click **Current LS**
- This solves the specified loading condition, **OK**
- Expand the **(+)General Postproc/(+)List Results** path. Select **Nodal Solution**
- Select **DOF Solution/Displacement Vector Sum, OK**
- This tabulates the x and y direction displacement values for each node in the model
- In case you want to save these results, you can create a screen shot using the Windows snipping tool, or save the image
- Select **Nodal Loads/All struc forc F, OK**
- This tabulates the x and y forces for each node in the model; make sure that applied forces show up correctly for node 6 and 8.
- In case you want to save these results, you can create a screen shot using the Windows snipping tool, or save the image
- For element forces select **General Postprocessing/ElementTable/Define Table / Add**. In the **Item, Comp, Results** table, choose **By Sequence Number**

and select **SMISC.** After **SMISC,** input 1. (The meaning of "1" is defined in the ANSYS user manual that can be accessed via the "Help" menu.) Then **Apply, Close,** etc. and under **Element Table / Plot Elem Table**, choose the **SMIS1** option
- Similarly, stress can also be solved for
- Click **(+)Plot Results** under the expanded **(+)General Postproc**. Select **Deformed Shape**
- Click **Def + Undeformed, 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.
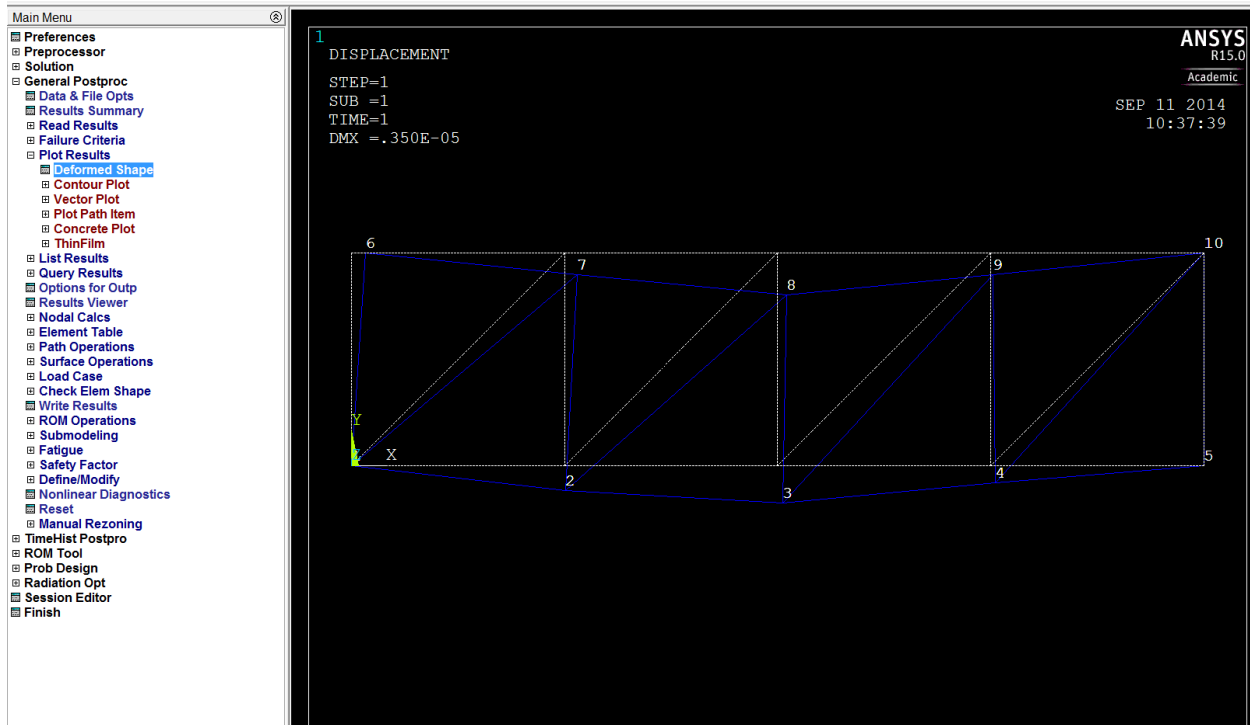

Figure 11: Visualization of Displacement Results

- Save & Exit: From **File**, select **Exit**, Click **Save Everything**
- To recall a saved database from a previous session:
  1. Open ANSYS Mechanical APDL
  2. From **File,** select **Resume Jobname**