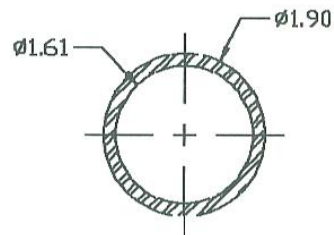


## CASE STUDY ANSYS EXAMPLE USING LINK180 ELEMENTS

The truss structure shown below is to be analyzed using ANSYS LINK180 elements. The frame has the following properties:

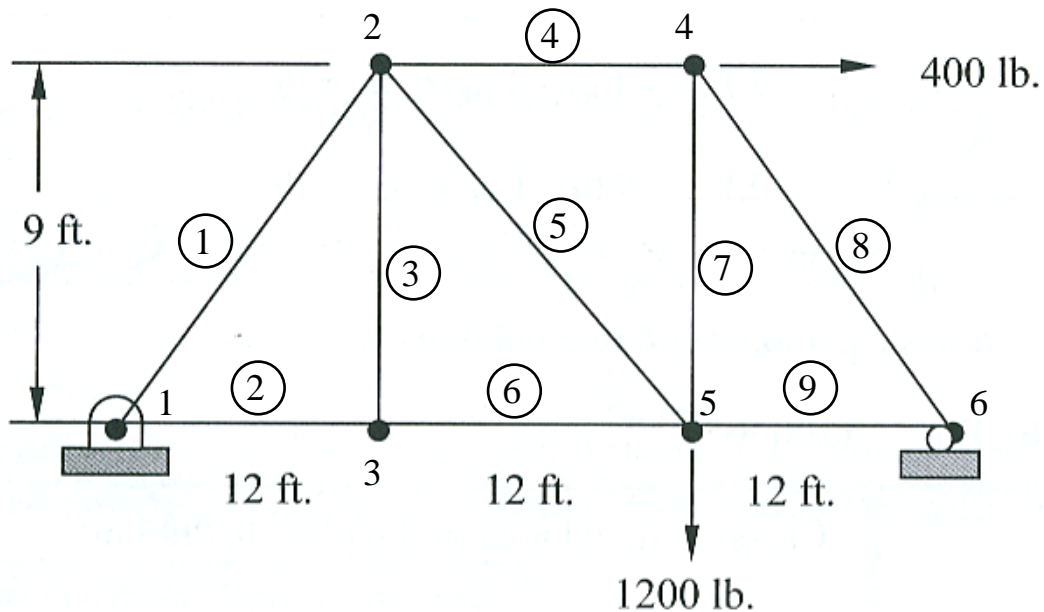
- Material: Steel pipe, NPS 1½", Schedule 40,  $E=30 \times 10^6$  psi
- Cross-sectional properties of the steel pipe:

$A = \text{cross-sectional area of the pipe} = 0.7995 \text{ in}^2$



Dimensions are in inches.

Determine the nodal displacements, the axial stress in each member, and the reactions.



## DESIGN OF COMPRESSION MEMBERS

	CENTRIC LOADING	ECCENTRIC LOADING
LONG COLUMNS	$\frac{l}{k} \geq \sqrt{\frac{2\pi^2 CE}{S_{yc}}}$ $\frac{P_{cr}}{A} = \frac{C\pi^2 E}{(l/k)^2}$ (Euler Equation)	$\frac{P_{cr}}{A} = \frac{S_{yc}}{1 + \left(\frac{ec}{k^2}\right) \sec \left[ \frac{1}{2} \left(\frac{l}{k}\right) \sqrt{\frac{P_{cr}}{AE}} \right]}$ (Secant Column Formula)  Valid for all $\frac{l}{k}$
INTERMEDIATE OR SHORT COLUMNS	$\frac{l}{k} < \sqrt{\frac{2\pi^2 CE}{S_{yc}}}$ $\frac{P_{cr}}{A} = S_{yc} - \left[ \frac{S_{yc}}{2\pi} \left(\frac{l}{k}\right) \right]^2 \frac{1}{CE}$ (Parabolic or J. B. Johnson Formula)	$\frac{P_{cr}}{A} = \frac{S_{yc}}{1 + \frac{ec}{k^2}}$  Valid only if $\frac{l}{k} \leq 0.282 \sqrt{\frac{AE}{P_{cr}}}$

where,

$$\frac{P_{cr}}{A} = \sigma_{cr} = \text{critical stress}$$

$$\frac{l}{k} = \text{slenderness ratio}$$

$$k = \sqrt{\frac{I}{A}} = \text{radius of gyration, } I = \text{moment of inertia about weak axis, } A = \text{cross-sectional area}$$

E=Young's Modulus

$S_{yc}$ =yield strength in compression

e=eccentricity

c=distance from centroidal axis to outer fiber in compression, and

C=end-condition constant. Values of C are given in the table below.

COLUMN END CONDITIONS	END-CONDITION CONSTANT, C		
	Theoretical Value	Conservative Value	Recommended Value <sup>1</sup>
Fixed-Free	0.25	0.25	0.25
Pinned-Pinned	1	1	1
Fixed-Pinned	2	1	1.2
Fixed-Fixed	4	1	1.2

<sup>1</sup>AISC (1989) recommended values, which include an additional factor of safety.

### **GENERAL COMMANDS IN ANSYS FOR TRUSS CASE STUDY**

1. Start ANSYS “Mechanical APDL 15.0”
2. Change working directory from default to your own working directory  
**Utility Menu > File > Change Directory**
3. Change jobname from default (default is “file”)  
**Utility Menu > File > Change Jobname**

**Note:** When working with your model, make sure to save the database often  
**Utility Menu > File > Save as Jobname.db, or**  
**Toolbar > Save Analysis (floppy disk icon), or**  
**Toolbar > SAVE\_DB**

4. For clarity, you may want to tell ANSYS what type of analysis you want to see menu choices for by using the “Preferences” command. Choose “Structural” for this problem to show only this discipline in the GUI.  
**Main Menu > Preferences → Structural**

### **PREPROCESSOR (PREP7 module)**

5. Define element type  
**Main Menu > Preprocessor > Element Type > Add/Edit/Delete → Add → 3D finit stn 180 (LINK180).** Use the defaults under “Options”
6. Define real constants (cross-sectional area in this case)  
**Main Menu > Preprocessor > Real Constants > Add/Edit/Delete → Cross-sectional area (=0.7995 in<sup>2</sup>, Real constant set #1)**
7. Define material properties, Young’s modulus and Poisson’s ratio. Note that Poisson’s ratio will not be used by the program since it is not needed for truss elements.  
**Main Menu > Preprocessor > Material Props > Material Models → Structural → Linear → Elastic → Isotropic (EX=30e6 psi, PRXY=0.3, Material Model Number 1)**
8. Create nodes. In this truss example, we can create the nodes directly because the model is simple, but in more complex models we will create first a geometry that can be meshed.  
**Main Menu > Preprocessor > Modeling > Create > Nodes > In Active CS** (requires nodal coordinate input). If you leave the node number blank in the menu, the program will default to the maximum node number used + 1. Make sure that the coordinates are defined in the proper consistent units (e.g., inches)
9. Define the elements using the nodes defined above. In this truss example, we can create the elements directly because the model is simple, but in more complex models we will create first a geometry that can be meshed.  
**Main Menu > Preprocessor > Modeling > Create > Elements > User Numbered > Thru Nodes → Pick element nodes with mouse or input numbers directly**
10. Apply displacement boundary conditions

**Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Nodes → Apply proper support conditions to nodes 1 and 6**

Note that the “Loads” menu can be found under the “Solution” menu as well  
**Main Menu > Solution > Define Loads**

11. Apply concentrated forces at nodes

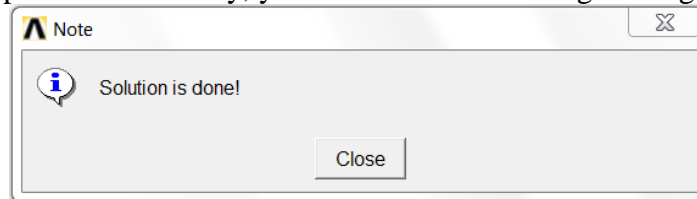
**Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Force/Moment > On Nodes → Apply forces of the proper magnitude and direction to nodes 4 and 5**

### **SOLVER (SOLUTION module)**

12. Run model

**Main Menu > Solution > Solve > Current LS**

If the solution completes successfully, you will see the following message



If you see a different message, it is likely that there was an error and the solver did not complete successfully. See the ANSYS Output Window for details if you get an error.

### **POST-PROCESSOR (POST1 module)**

13. Post-process and view results

**Main Menu > General Postproc > Plot results > Deformed Shape**

You can animate the deformed shape for visual confirmation

**Utility Menu > PlotCtrls > Animate > Deformed Shape** (you can control the speed of the animation with the slider bar in the pop-up window)

**Main Menu > General Postproc > Plot results > Contour Plot > Element Solu → Structural Forces → X-Component of force**

For truss elements, you must first define an element table in order to plot the axial stress results (note that the axial stress in a truss member is simply related to the axial force by  $\sigma = F/A$ ):

**Main Menu > General Postproc > Element Table > Define Table → Add → By sequence number → LS, 1** (you can define the label “Lab” for this item as SIGMA\_X for identification). The value of LS, 1 comes from Table 180.2 from the element description in “Help Topics → LINK180”. A value of LS, 2 will produce the same results since the stress is constant along the element (LS, 1 and LS, 2 refer to the axial stress values at nodes i and j in the element).

You can list the values for the table you have defined in order to double check:

**Main Menu > General Postproc > Element Table > List Element Table → SIGMA\_X**

In order to plot the axial stress values use:

**Main Menu > General Postproc > Plot Results > Elem Table → SIGMA\_X** (make sure that you choose “No - do not avg” for Avglab option, otherwise the stress values from connecting elements will be averaged)

For a listing of results, you can use

Forces

**Main Menu > General Postproc > List Results > Element Solution → Structural Forces → X-Component of force**

Force and Axial Stress

**Main Menu > General Postproc > List Results > Element Solution → Line Element Results → Element Results**

Axial Stress from defined Element Table, SIGMA\_X

**Main Menu > General Postproc > Element Table > List Element Table → SIGMA\_X**

Alternatively, you can use the “Results Viewer” for all post-processing activities (except element table results). The Results Viewer is a compact toolbar for viewing your analysis results. Selecting the Results Viewer disables much of the standard GUI functionality. Many of these operations are not available because of PowerGraphics limitations. However, a good deal of the POST1 functionality is contained in the Result Viewer menu structure, and in the right and middle mouse button context sensitive menus that are accessible when you use the Results Viewer. You can access the Results Viewer from

**Main Menu > General Postproc > Results Viewer**

14. Before exiting, always save a copy of the command log file from the database log as a backup:

**Utility Menu > File > Write DB Log File → Jobname.lgw**