

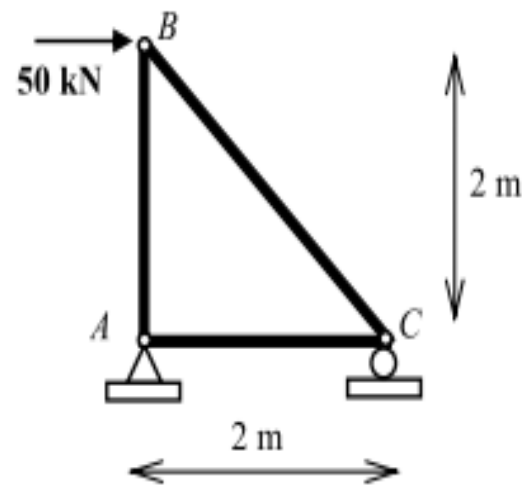
# CE327-Analysis and Design Software Lab

SAP2000 Demonstration

- The following is a step-by-step procedure for analysis a two-dimensional truss structure using SAP 2000. The order of some of these steps is not critical; however, all step should be completed before execution of the analysis.
- The following tutorial will focus on determining the forces in each member of the truss shown below. Assume all members are pin connected.

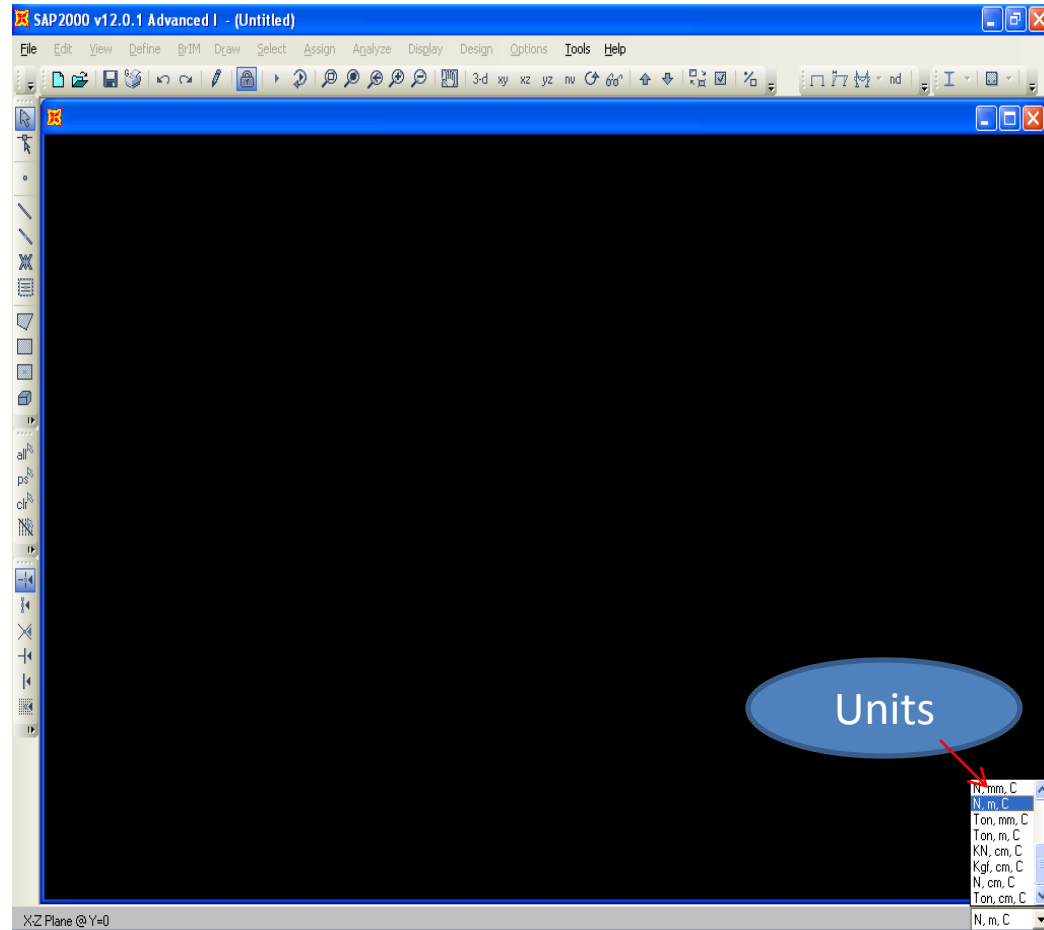
Given problem:

1. Find out the end forces and end deformations of all the members for the 2-D truss system shown in **Figure 1** using the *direct stiffness method*. For all members, consider,  $E = 200 \text{ GPa}$ ,  $A = 500 \text{ mm}^2$ ,  $I = 26041.67 \text{ mm}^4$ .



**Figure 1**

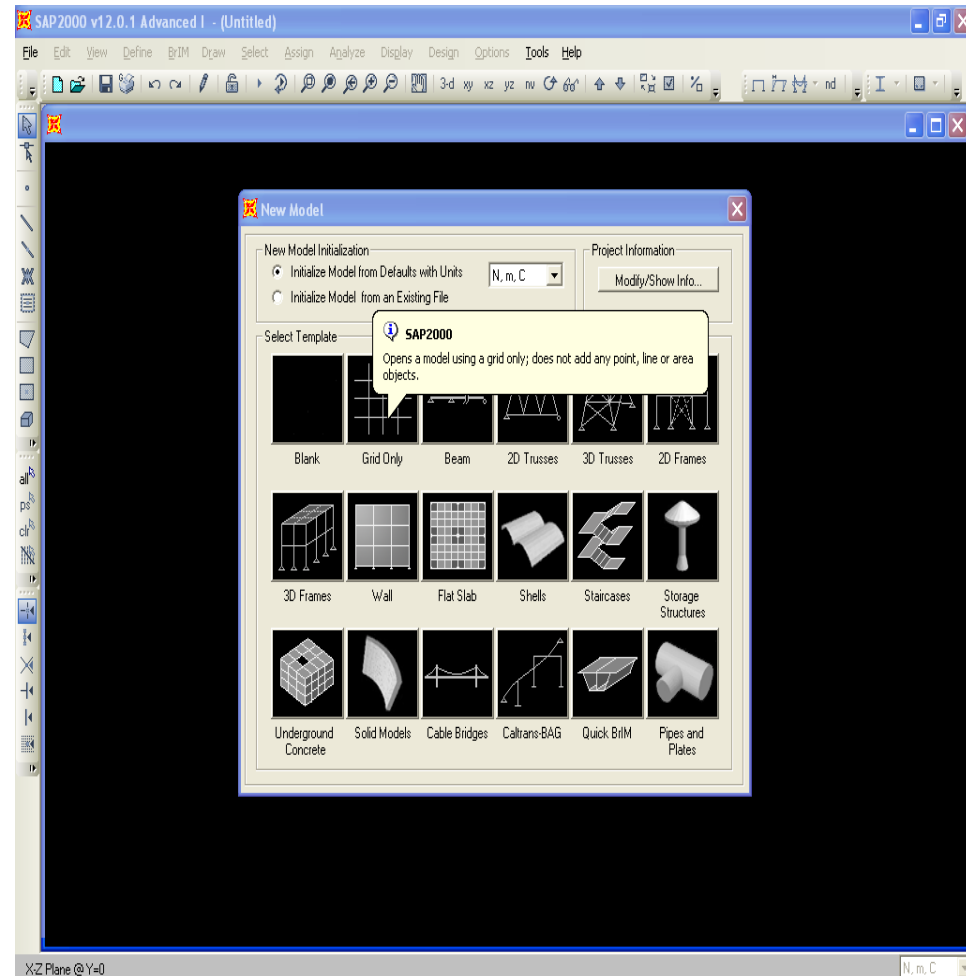
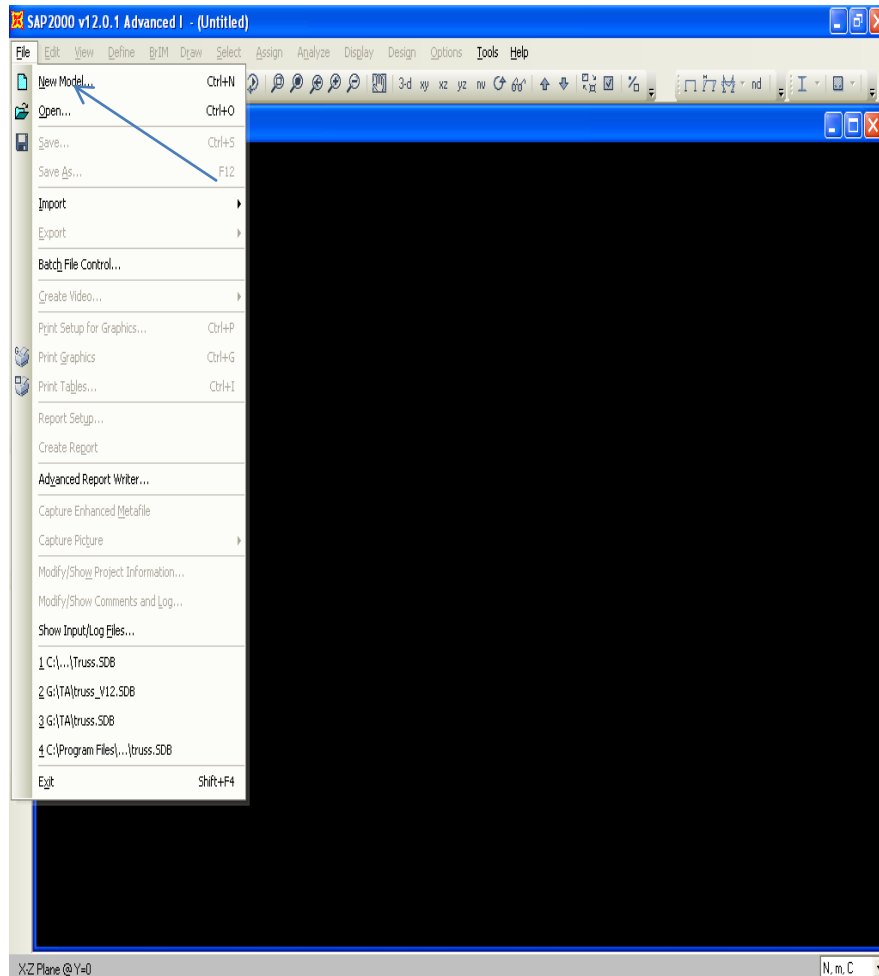
When you start SAP 2000 Version 12 you should see the following interface window:



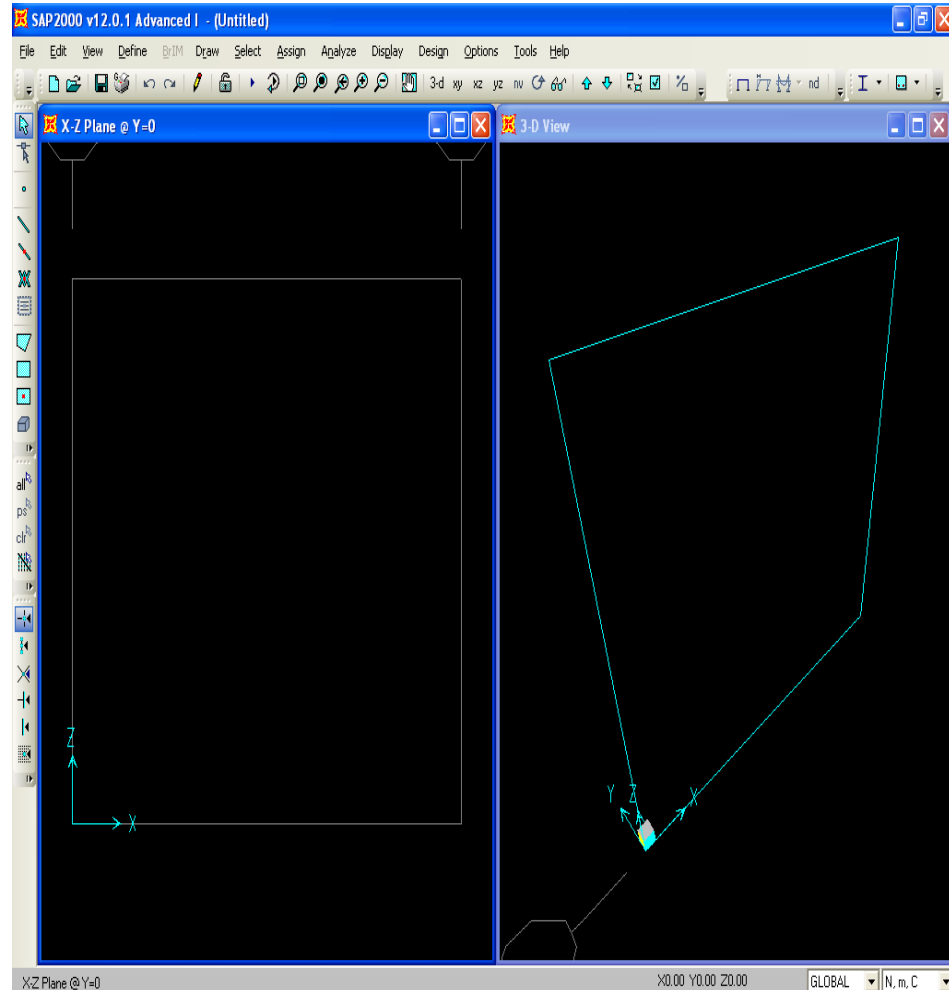
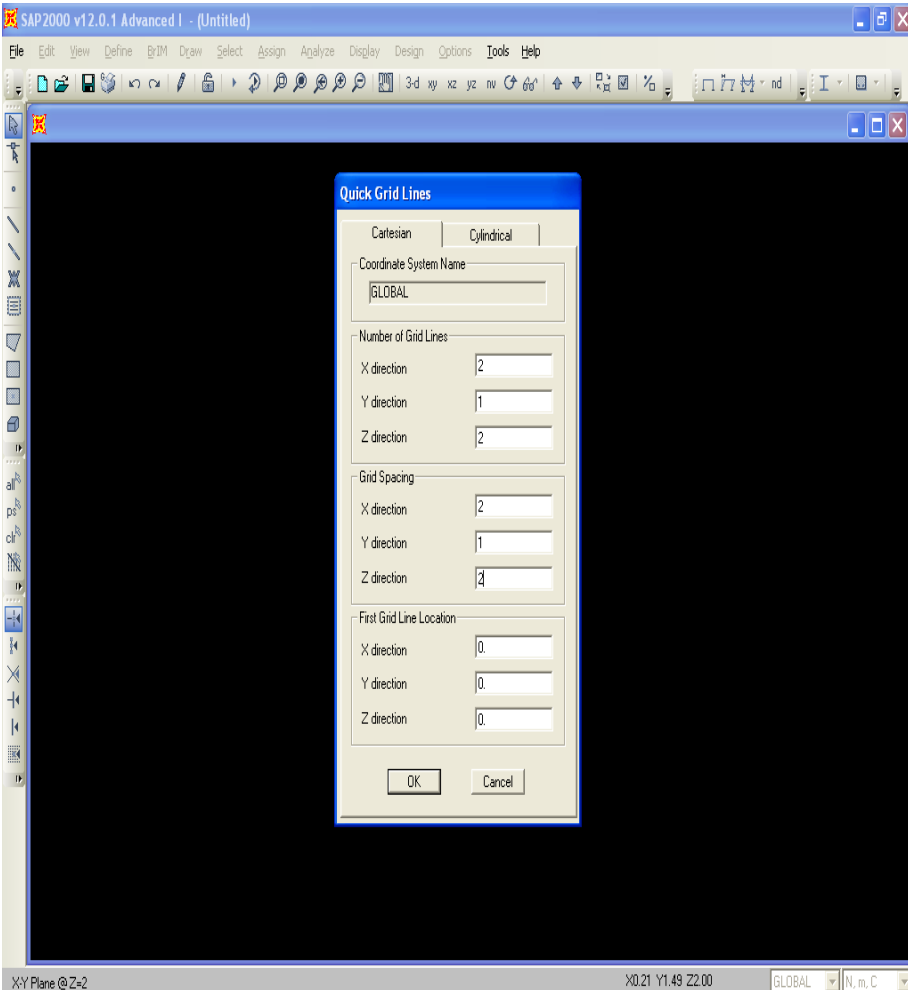
**Step 1: Set Problem Dimensions** - On the bottom on the interface window, set the desired units for the problem using the pull-down menu. In this example, the units are Newton and meter.

## Step 2:

A) When you select new model on the menu, window will appear showing different templates such as , beam, 2D & 3D trusses, 2D frames etc. For our problem we can define it by two ways, 1) by directly taking given template for 2D truss 2) & by **using grid only** template.

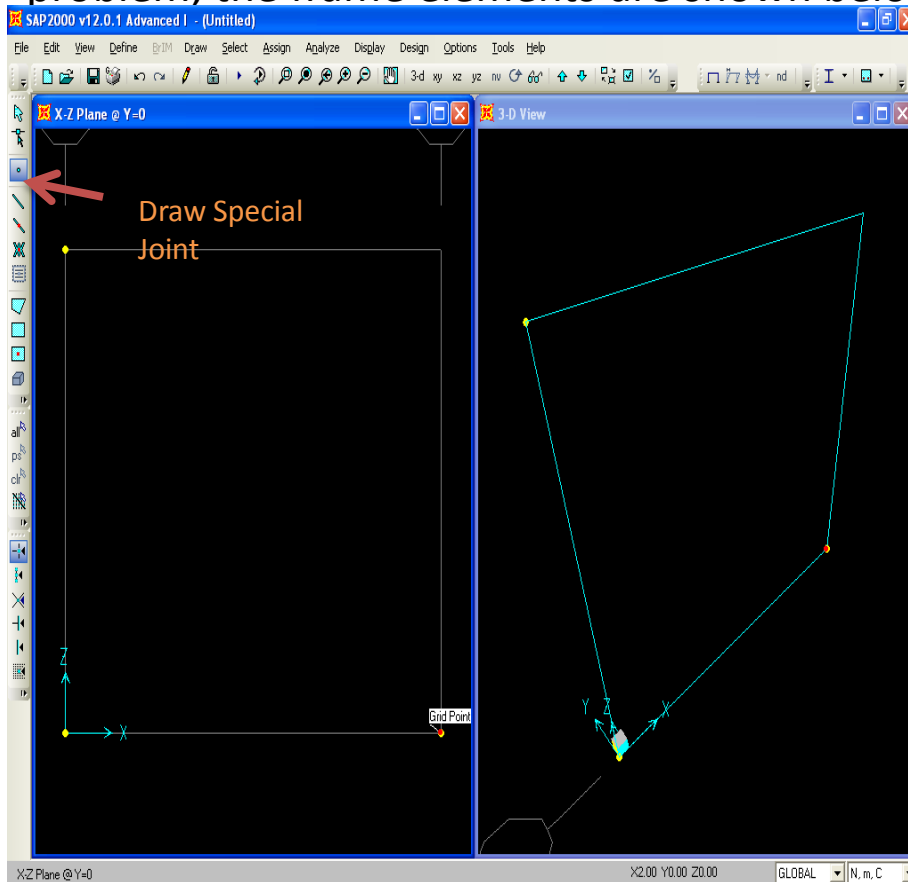


- B)** For this problem we will do it by using grid only template. Determine the appropriate number of grid line and grid spacing to locate the joints of the truss.
- C)** After clicking Ok button, By default SAP 2000 show two views of your problem, typically a 3-D view and an x-y plane view. To adjust the views, select an window and click on the appropriate view button located along the top edge of the interface window.

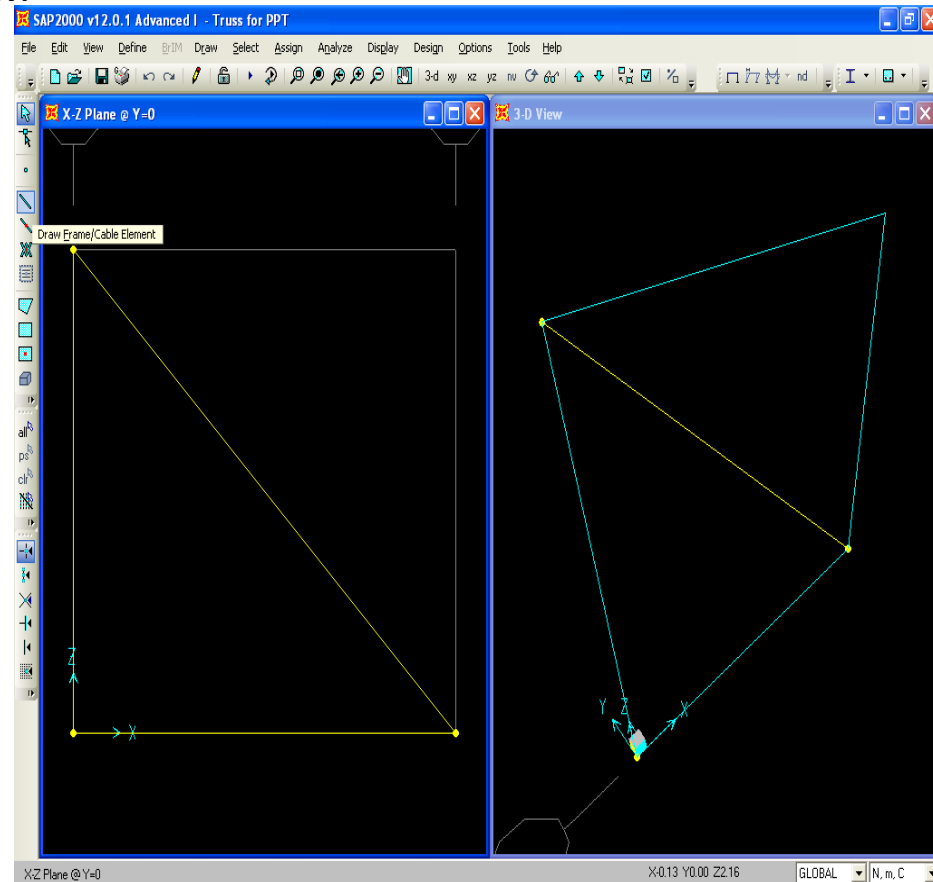


**Step 3: Locate Truss Joints** - To define the joint locations, select the **Draw Special Joint** button on the left tool bar (or go to draw>draw special joint). Click on grid intersection lines to define joints. For this problem the joint locations are shown below.

**Step 4: Draw Frame Elements** - To define each frame element, select the **Draw Frame Element** button on the lower tool bar. To define an element, click on a joint at the beginning of the element and then on the joint at the end of the element. For this truss problem, the frame elements are shown below.



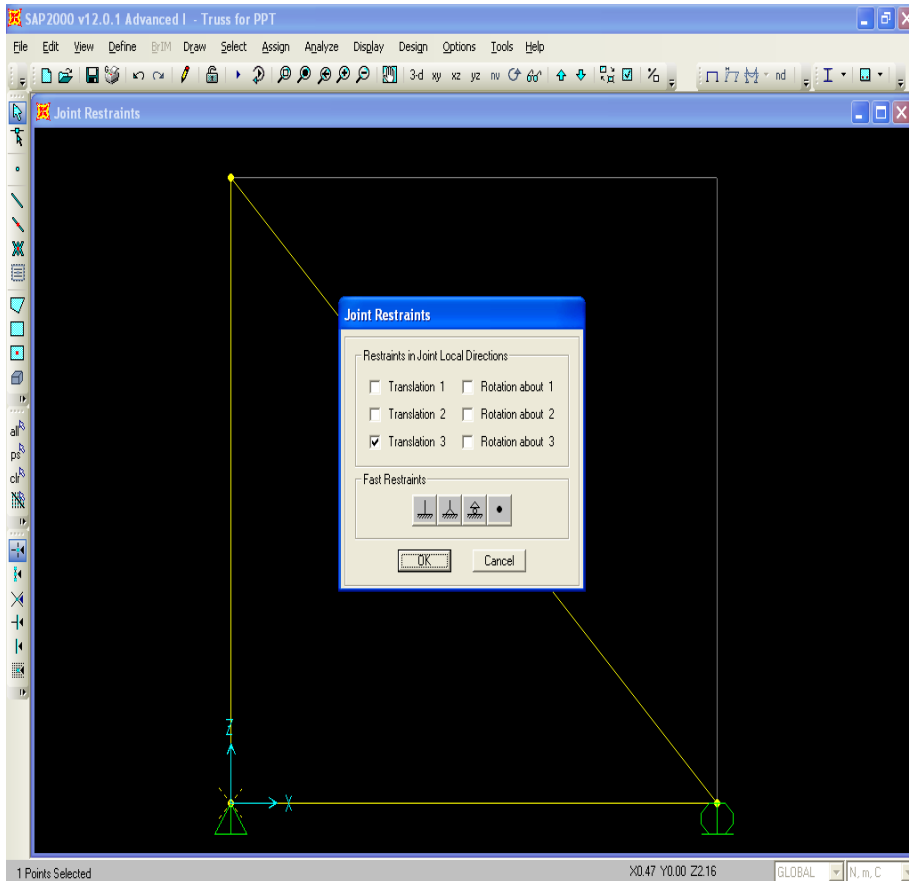
Step 3



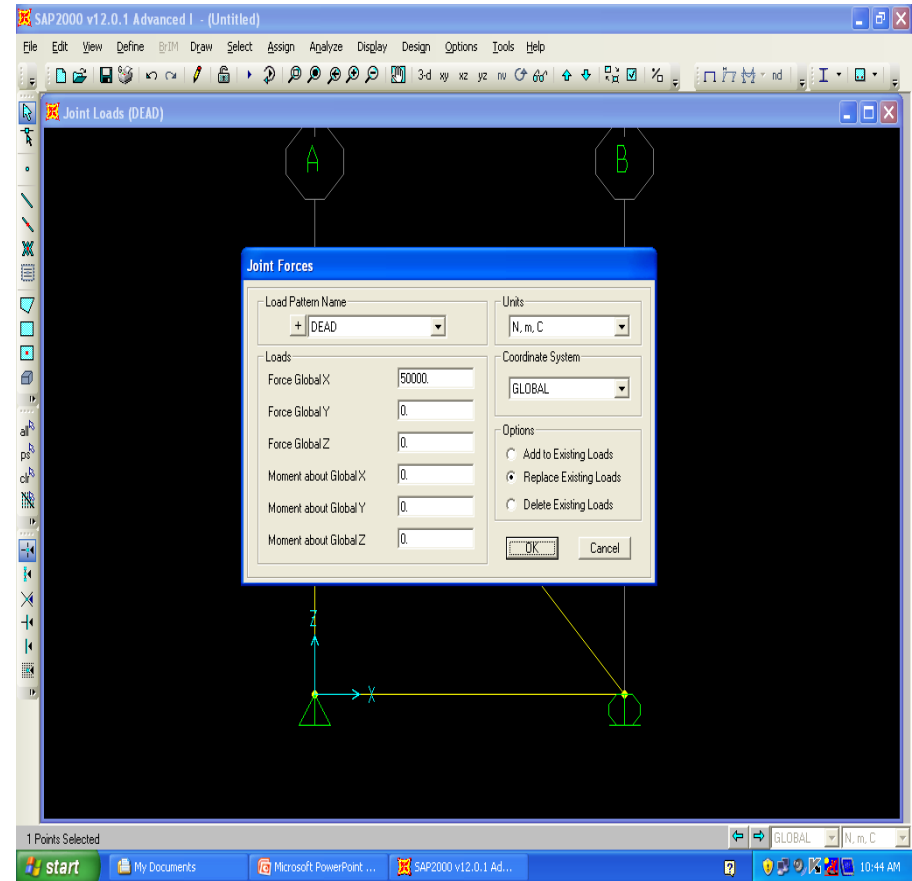
Step 4

**Step 5: Define Structural Supports** - To define the location and type of structural support, select the support location by clicking on the joint with the pointer. A yellow "X" should appear at the joint to indicate that it is currently selected. Next click on the Joint Restraint button (go to assign>joint>joint restraints) & then assign the support given in the problem.

**Step 6: Apply Forces at Joints** - To apply forces at a joint, select the joint with the pointer and click on the Assign Joint Loadings button & put 50KN in the x direction.



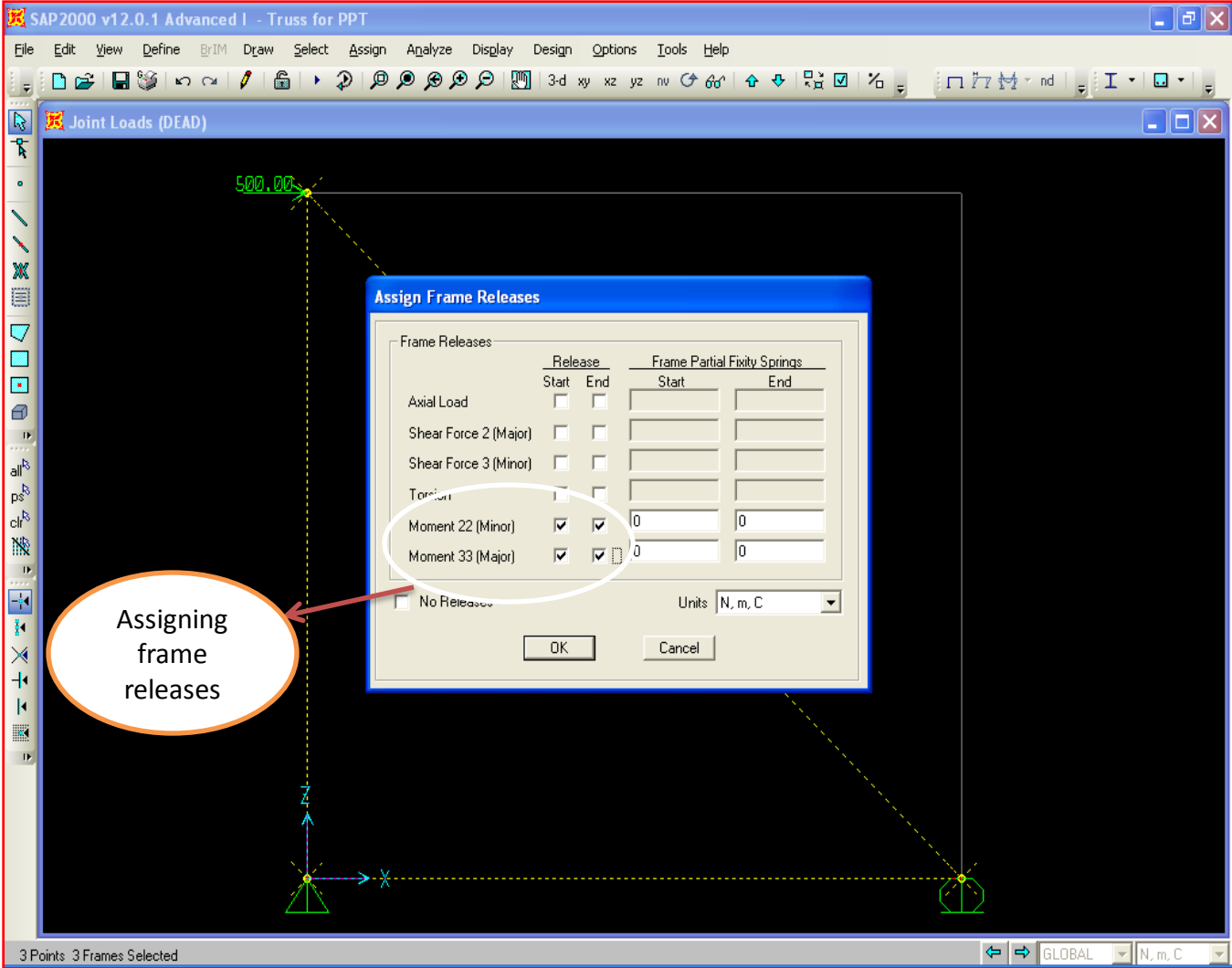
Step 5



Step 6



**Step 7: Release Internal Moments at Joints - SAP 2000** assumes that all structures are frames. Therefore, to analyze a truss structure we should convert each joint from a fixed connection to a pin connection. To ensure that every joint in the structure is pin connected, select all the members by clicking the Select All button on the left tool bar. Next click on Assign menu >Frame > Releases /partial fixity and then Frame Releases window will appear.



## Step8:Define & Assign

### Material Properties –

A)To define the properties of a material , select the Define menu located along the top the SAP 2000 interface window and then click on Materials. The Define Materials window will appear .On this menu you can change the properties of materials. In this example, select add new material and change the properties as shown. And go to switch to advanced property display.

The screenshot displays the SAP2000 v12.0.1 Advanced I - truss\_V12 interface. The main window shows a 3D view of an X-Z Plane @ Y=0. Two dialog boxes are open:

- Define Materials:** A list of materials is shown, with 'STEEL' selected. The 'Click to:' section contains buttons for 'Add New Material Quick...', 'Add New Material...', 'Add Copy of Material...', 'Modify/Show Material...', and 'Delete Material'. The 'Show Advanced Properties' checkbox is checked.
- Material Property Data:** A dialog box for defining material properties. The 'General Data' section includes 'Material Name and Display Color' (STEEL), 'Material Type' (Steel), and 'Material Notes' (Modify/Show Notes...). The 'Weight and Mass' section includes 'Weight per Unit Volume' (0), 'Mass per Unit Volume' (0), and 'Units' (N, m, C). The 'Isotropic Property Data' section includes 'Modulus of Elasticity, E' (2.000E+11), 'Poisson's Ratio, U' (0.3), 'Coefficient of Thermal Expansion, A' (1.170E-05), and 'Shear Modulus, G' (7.692E+10). The 'Other Properties for Steel Materials' section includes 'Minimum Yield Stress, Fy' (2.500E+08), 'Minimum Tensile Stress, Fu' (2.500E+08), 'Effective Yield Stress, Fye' (2.500E+08), and 'Effective Tensile Stress, Fue' (2.500E+08). The 'Switch To Advanced Property Display' checkbox is checked.

The status bar at the bottom shows 'X-1.78 Y0.00 Z4.01' and 'GLOBAL'.

**B) Change the value in the Weight per unit Volume & mass per unit volume input field to zero. For this example problem, put the given values of Modulus of elasticity, Poisson's ratio & yield stress(250Mpa). Now we have a material named STEEL ,change the nonlinear material data as shown in figure. Then select all>assign>material property overwrites>STEEL.**

The screenshot shows the SAP2000 software interface with two dialog boxes open for material definition.

**Material Property Data Dialog:**

- Material Name: STEEL
- Material Type: Steel
- Symmetry Type: Isotropic
- Modulus of Elasticity (E): 2.000E+11
- Poisson's Ratio (U): 0.3
- Coeff of Thermal Expansion (A): 1.170E-05
- Shear Modulus (G): 7.692E+10
- Weight and Mass: Weight per Unit Volume: 0, Mass per Unit Volume: 0
- Units: N, m, C
- Other Properties for Steel Materials:
  - Minimum Yield Stress, Fy: 2.500E+08
  - Minimum Tensile Stress, Fu: 2.500E+08
  - Effective Yield Stress, Fye: 2.500E+08
  - Effective Tensile Stress, Fue: 2.500E+08
- Advanced Material Property Data buttons: Nonlinear Material Data..., Material Damping Properties..., Time Dependent Properties..., Thermal Properties...

**Nonlinear Material Data Dialog:**

- Material Name: STEEL
- Material Type: Steel
- Hysteresis Type: Kinematic
- Stress-Strain Curve Definition Options:
  - Parametric
  - User Defined
- User Stress-Strain Curve Data:
  - Number of Points in Stress-Strain Curve: 3
  - Table:

	Strain	Stress	Point ID
1	-1.250E-03	-2.500E+08	
2	0.	0.	A
3	1.250E-03	2.500E+08	

Additional controls in the Nonlinear Material Data dialog include a plot area, 'Order Rows', 'Show Plot...', 'OK', and 'Cancel' buttons.

**Step 9: Define Frame Sections** - To define the cross-section properties of a structural element click on the Define menu located along the top the SAP 2000 interface window and then click on Frame Sections. We can Define sections in 3 different ways ,

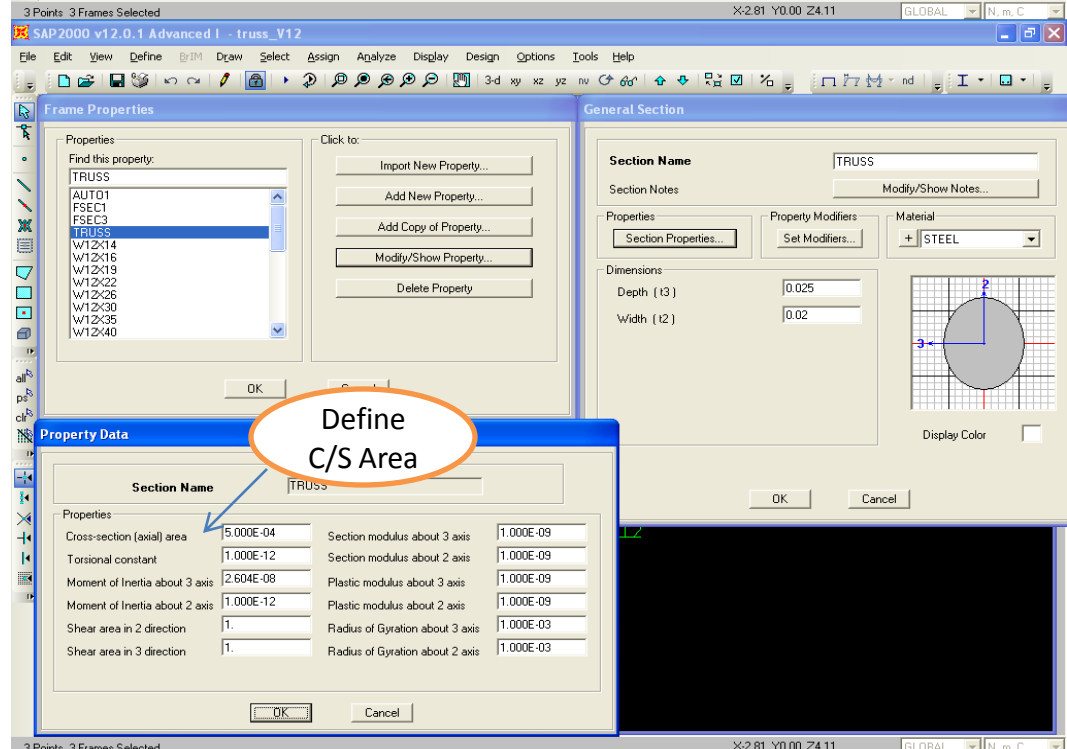
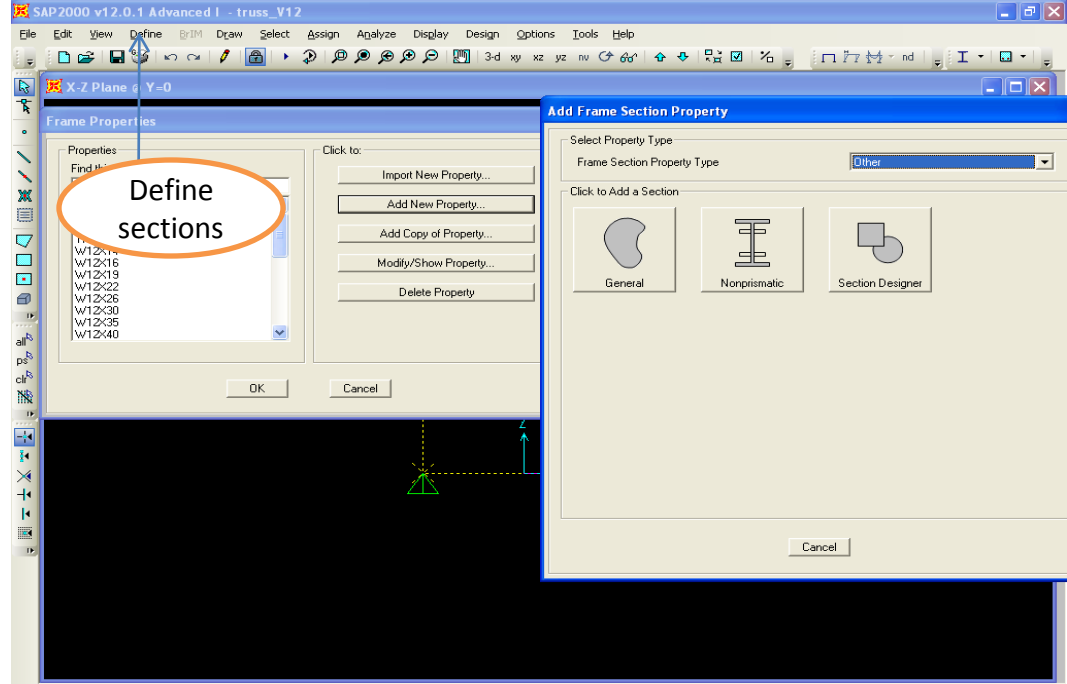
a) Define>Section properties>Frame sections>Import new property>I/Wide flange>sections.pro

b) Define>Section properties>Frame sections>Add new property>Frame section property type(other)>general>fill section properties(Area, Moment of inertia. etc)>Enter depth & Width

c) Define>Section properties>Frame sections>Add new property>Frame section property type(other)>general>Section designer

For this problem we will use **option b)** to define sections.

**Step10:Assign Frame sections-** To assign the frame section , first select all member then go to Assign>Frame>Frame sections>Frame properties and then click ok.



**Step 11: Set Analysis Options and Run Analysis** - In this example, the truss structure is modeled in the x-z plane. To limit analysis to variables in the x-z plane click on the Analyze menu located along the top the SAP 2000 interface window and then click on Set Options. The truss structure is now ready for analysis. To analyze the model press the Run Analysis button. After the Analysis Complete window has been closed, typically SAP 2000 displays the deflected shape of the structure.

The image shows two screenshots from the SAP 2000 v12.0.1 Advanced I software interface. The left screenshot displays the 'Analysis Options' dialog box, which is used to configure the analysis parameters. The 'Available DOFs' section has checkboxes for UX, UY, UZ, RX, RY, and RZ. The 'Fast DOFs' section has four options: Space Frame, Plane Frame, Plane Grid, and Space Truss. The 'XZ Plane' option is selected. The 'Tabular File' section has a checkbox for 'Automatically save Microsoft Access or Excel tabular file after analysis'. A callout bubble points to the 'XZ Plane' option with the text 'Select X-Z plane'. The right screenshot displays the 'Set Load Cases to Run' dialog box, which is used to select the load cases to be analyzed. The dialog box contains a table with columns for Case Name, Type, Status, and Action. The 'DEAD' and 'MODAL' cases are listed. A callout bubble points to the 'MODAL' case with the text 'Set load cases to run'. The 'Run/Do Not Run All' button is highlighted.

**Analysis Options**

Available DOFs

UX  UY  UZ  RX  RY  RZ

Fast DOFs

Space Frame Plane Frame Plane Grid Space Truss

XZ Plane XY Plane

Tabular File

Automatically save Microsoft Access or Excel tabular file after analysis

File name: \_\_\_\_\_

Group: \_\_\_\_\_

**Set Load Cases to Run**

Case Name	Type	Status	Action
DEAD	Linear Static	Not Run	Run
MODAL	Modal	Not Run	Do Not Run

Click to:

Run/Do Not Run Case  
Show Case...  
Delete Results for Case  
Run/Do Not Run All  
Delete All Results  
Show Load Case Tree...

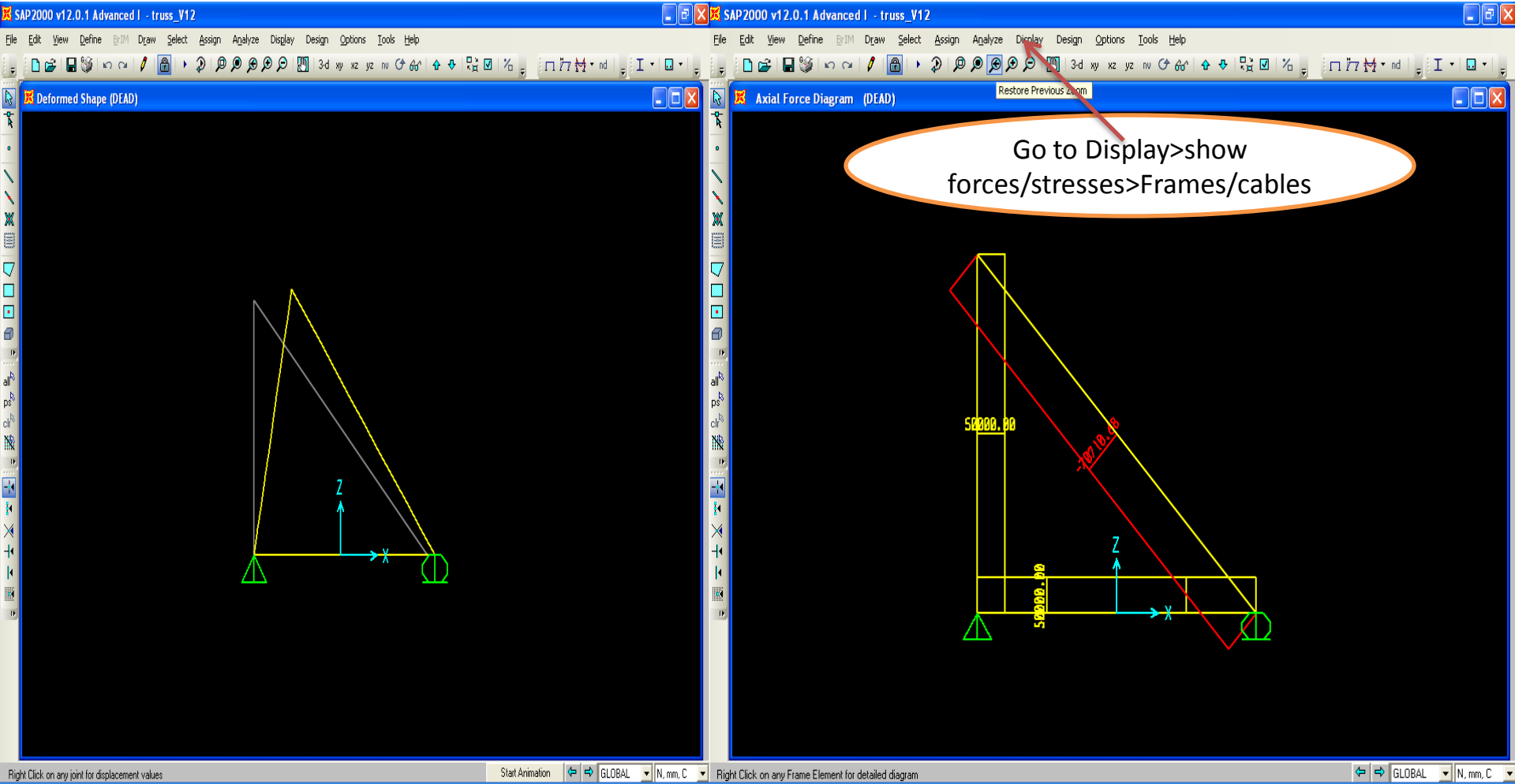
Analysis Monitor Options

Always Show  
 Never Show  
 Show After 4 seconds

Model-Alive  
Run Now  
OK Cancel

3 Frames Selected X0.65 Y0.00 Z2.80 GLOBAL N, m, C

**Step 12: Print Truss Forces** - To get a quick feel for the relative magnitude of the forces in the truss, select the Member Force Diagram for Frames button along the top tool bar.



Similar procedure you can follow for the following problems,

