

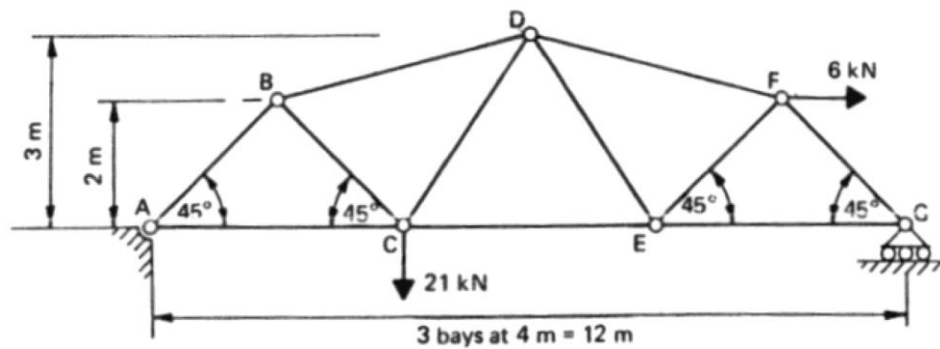
Pin Jointed Truss Analysis

Title

Statically determinate structural analysis for a pin jointed Truss

Description

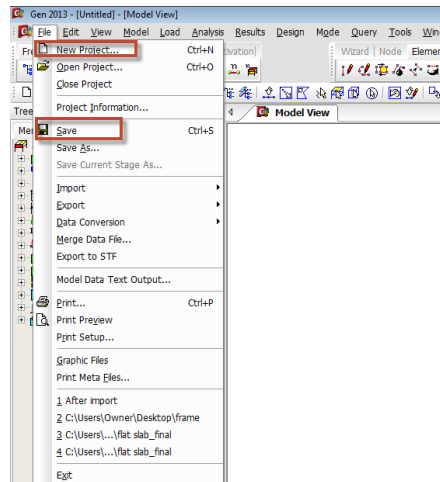
The figure below shows a pin jointed framework supporting a vertical and horizontal load. Determine the three reactive components needed to maintain overall equilibrium and the magnitude of the forces in members FD, FE, FG and CE, stating whether the forces are tensile or compressive.



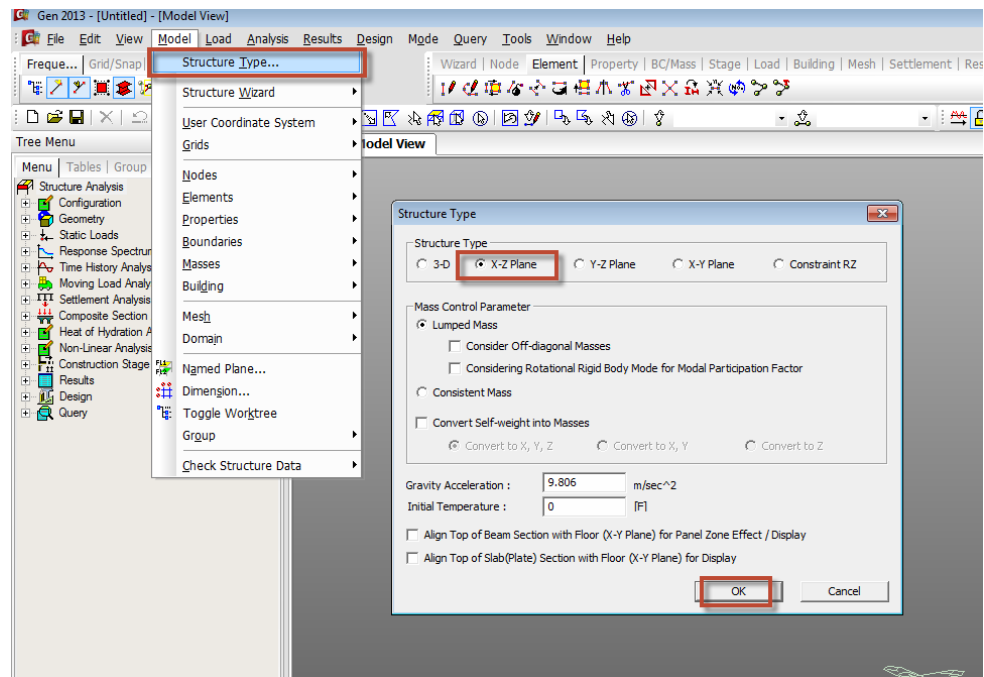
Structural geometry and analysis model

Finite Element Modelling:

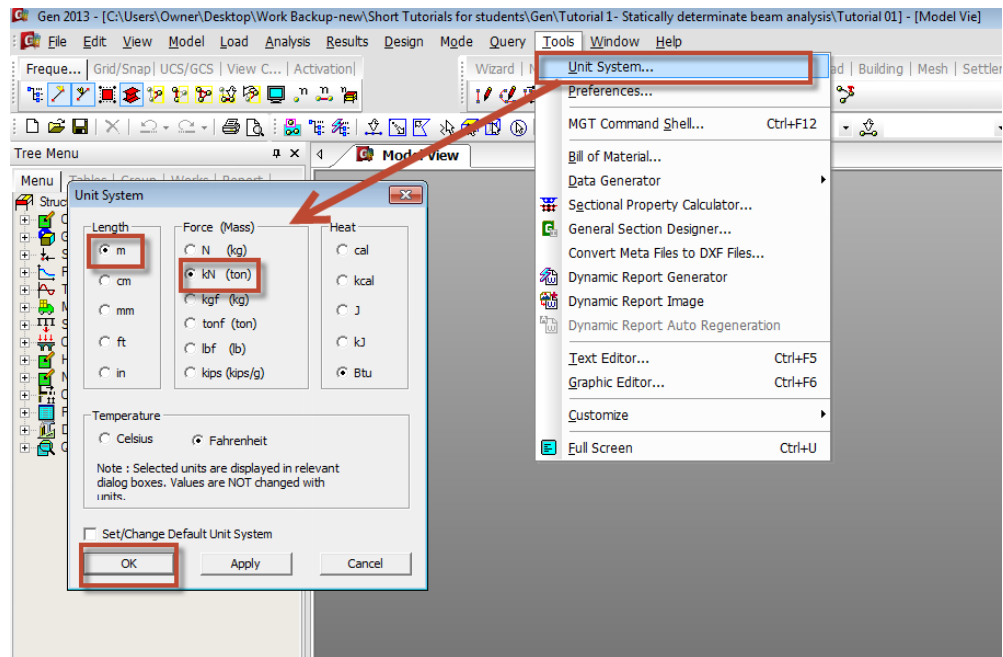
- **Analysis Type:** 2-D static analysis (X-Z plane)
Step 1: Go to **File>New Project** and then go to **File>Save** to save the project with any name



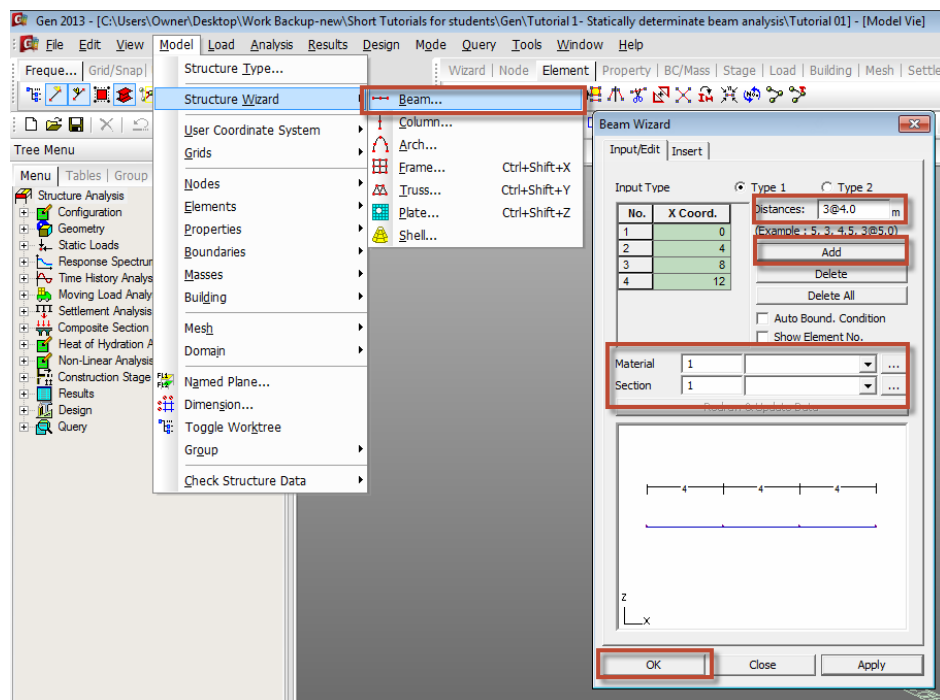
Step 2: Go to **Model>Structure Type** to set the analysis mode to 2D (X-Z plane)



- **Unit System:** kN,m
Step 3: Go to **Tools>Unit System** and change the units to kN and m.

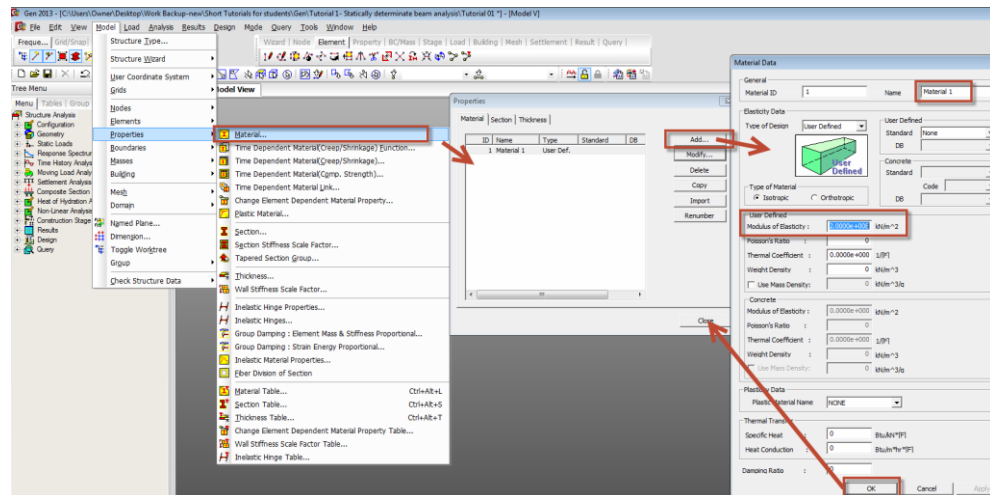


- **Dimension:** Length= 3@4.0m= 12.0m
- Step 4:** Go to **Model>Structure Wizard>Beam** and type in 3@4.0 in the Distances box. Press Add. Type 1 in the Material and Section ID entry.



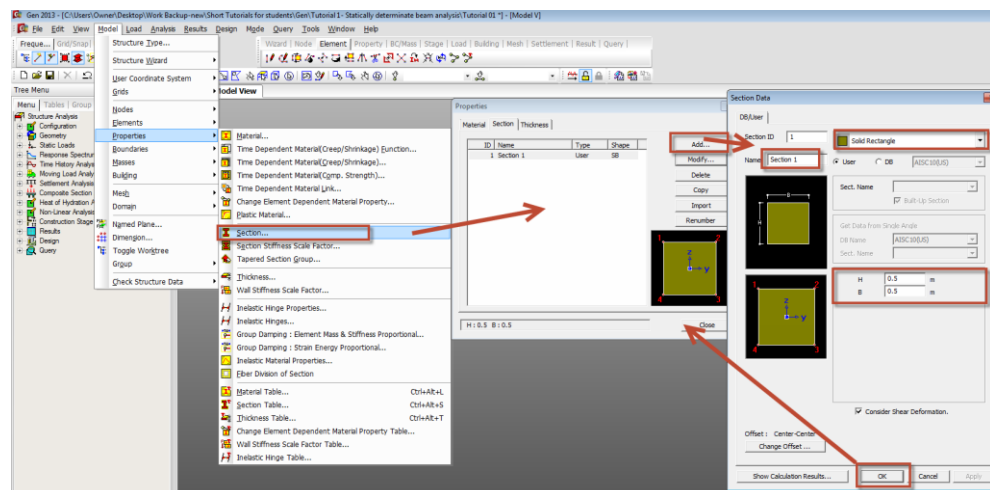
- **Element type:** Beam element (Beam Wizard from Step 4 generates beam elements automatically)
- **Material:** Modulus of elasticity, $E = 2.0 \times 10^8 \text{ kN/m}^2$

Step 5: Go to **Properties>Material Properties>Add**. Select User defined in the Type of Design and Enter $E=2.0 \times 10^8 \text{ kN/m}^2$. Enter a name for the material and click OK and Close.

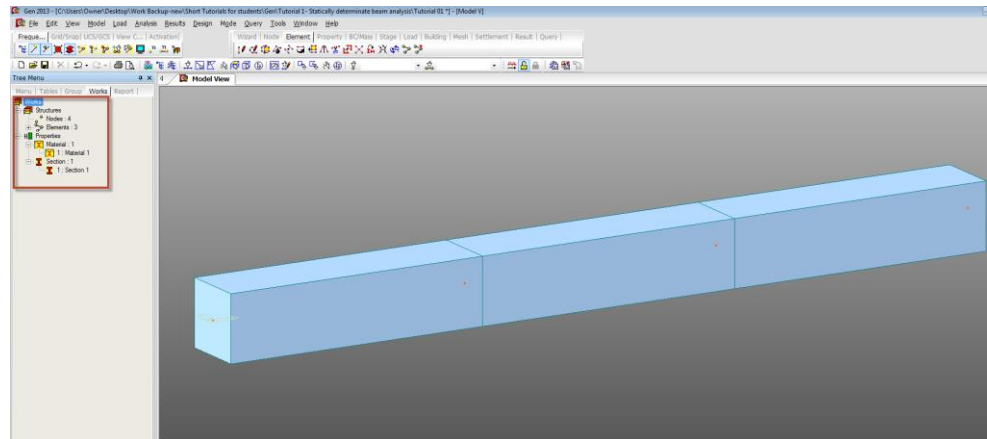





- **Section Property:** $B \times H = 0.5 \text{ m} \times 0.5 \text{ m}$

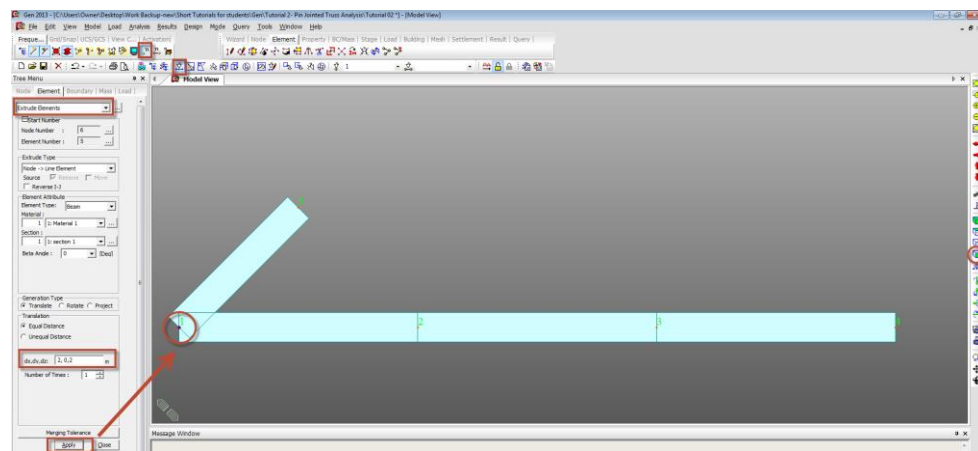
Step 6: Go to **Properties>Section Properties>Add**. Select Solid Rectangle in the Type of Section Drop down menu and check on User. Enter $H=0.5$ and $B=0.5$. Enter a name for the Section.

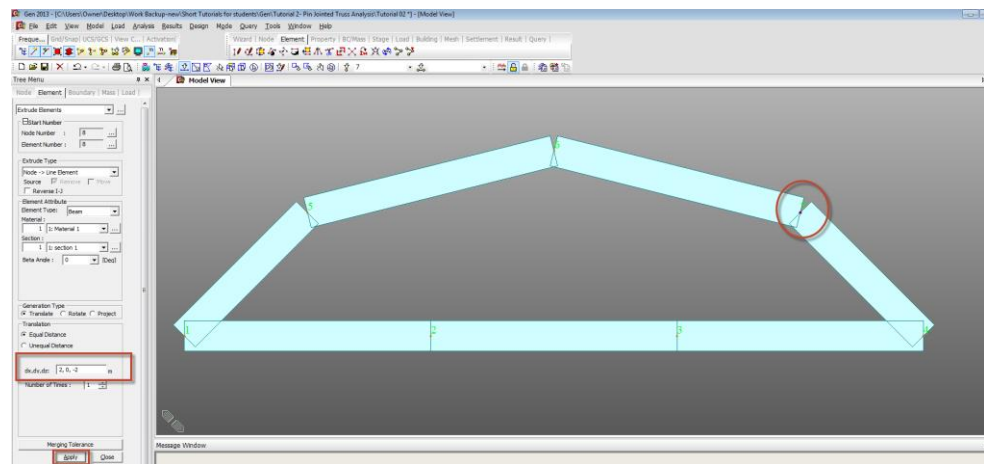
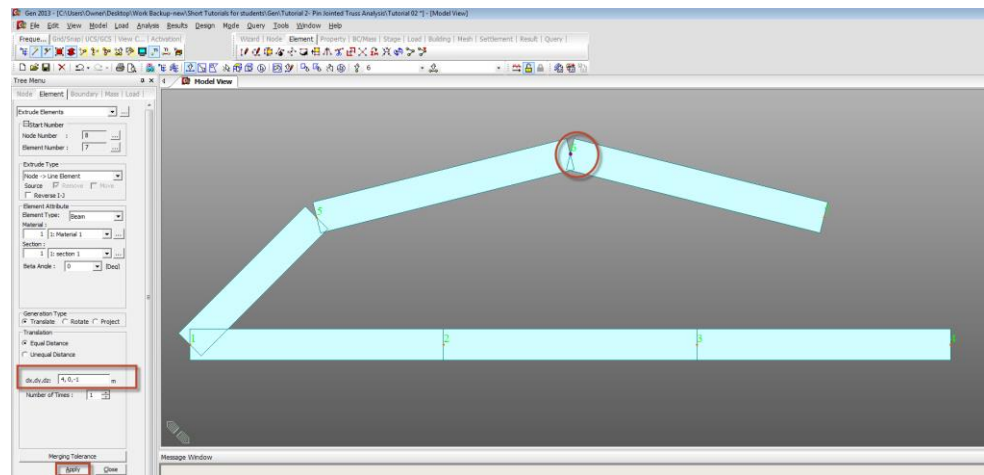
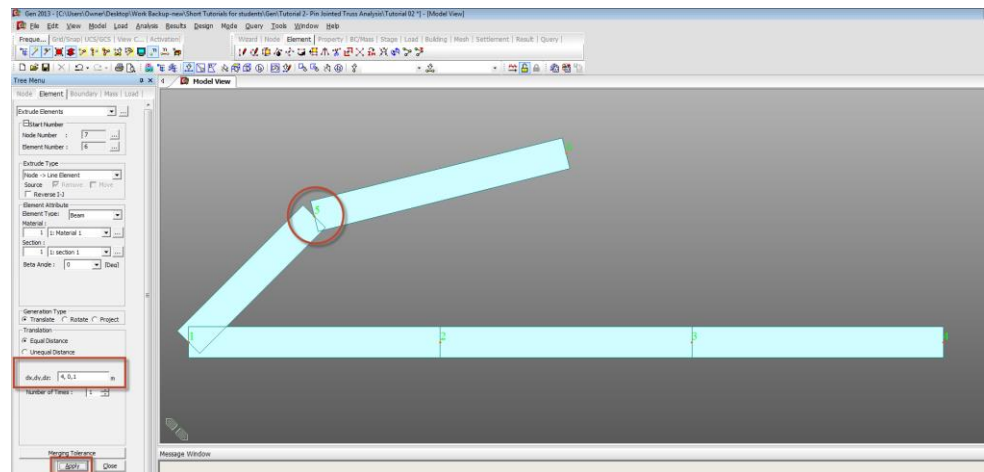



You can see the shape of the section in the model generated. Go to the Works Tree to check the information for your model.



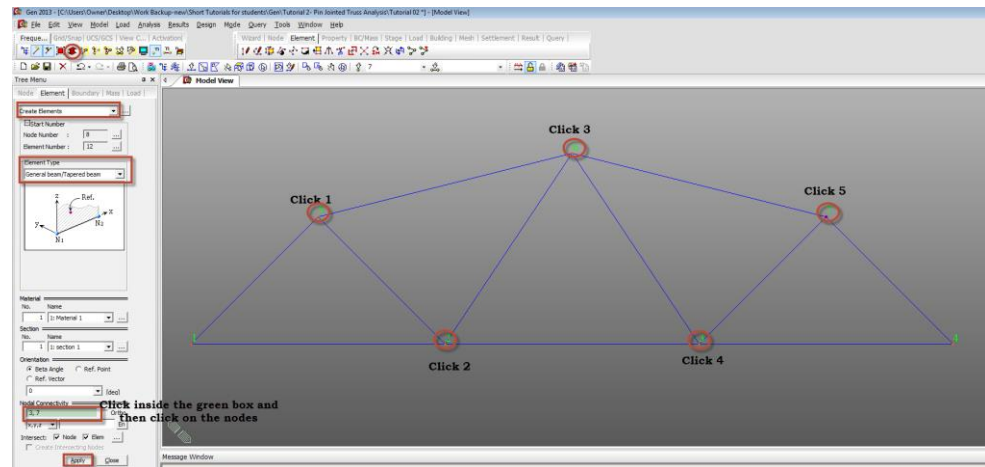
Step 7: Display the node numbers by clicking on . Display the front view by clicking on the  option on the right. Go to **Model>Elements>Extrude Elements** and select the node number 1 using select single option . Enter 2,0,2 in the (dx,dy,dz) box and click Apply. Repeat the process by selecting node 5 and entering (4,0,1) and Apply. Then again select node 6, enter (4,0,-1) and Apply. Finally select node 7, enter (2,0,-2) and Apply.





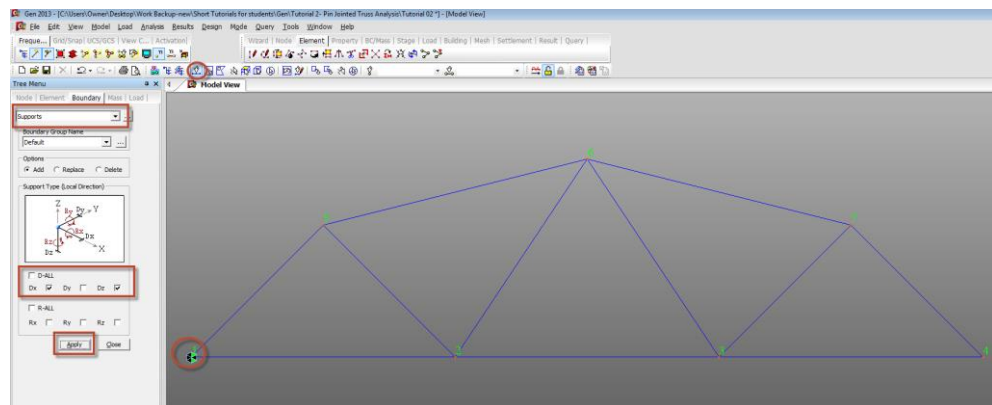
Step 8: Switch off Hidden View by clicking on . Then go to **Model>Elements>Create Element** and select element type as General Beam element. Left Click on the Node Connectivity box and connect the nodes or snap directly on the nodes in the main model

view. See figure below.




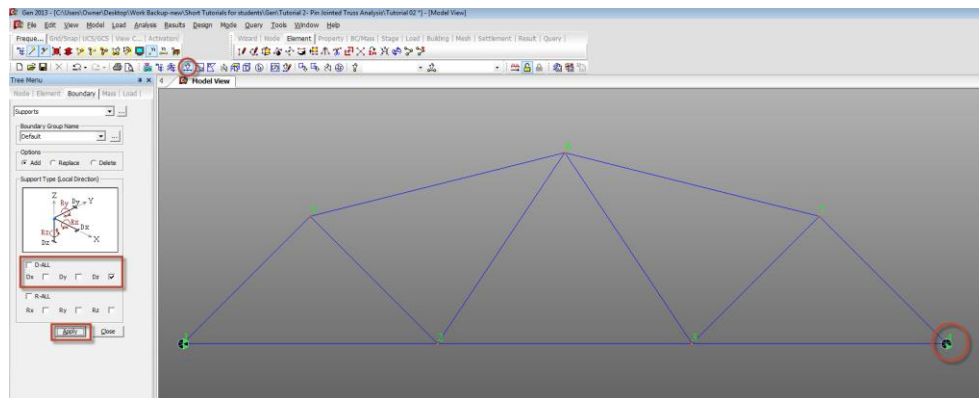
- **Boundary Condition:** Simply Supported (Pinned at A and Roller at G)

Step 9: Use select single  to select or highlight node 1 as shown in figure below. Go to **Model>Boundaries>Supports** and check on Dx and Dz and Apply.




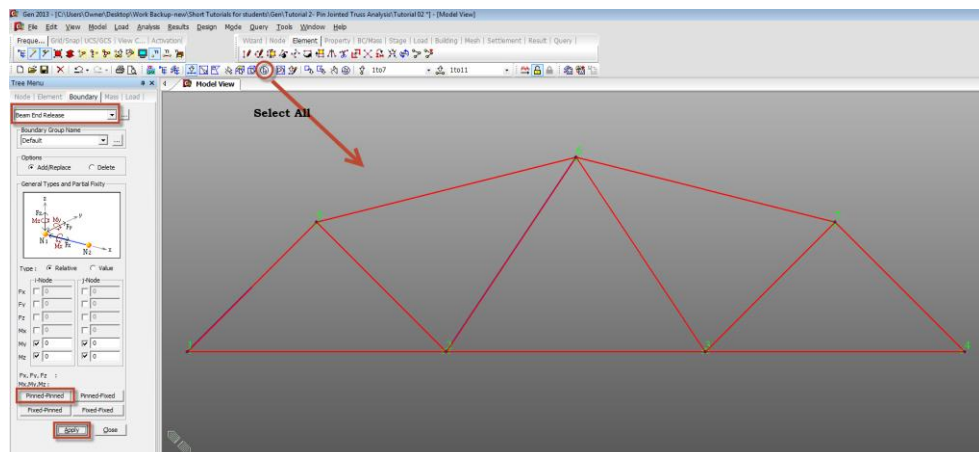
This becomes the Pinned support at A

Step 10: Again use Select Single , this time to highlight or select node 4 and check on Dz only, Apply and Close.



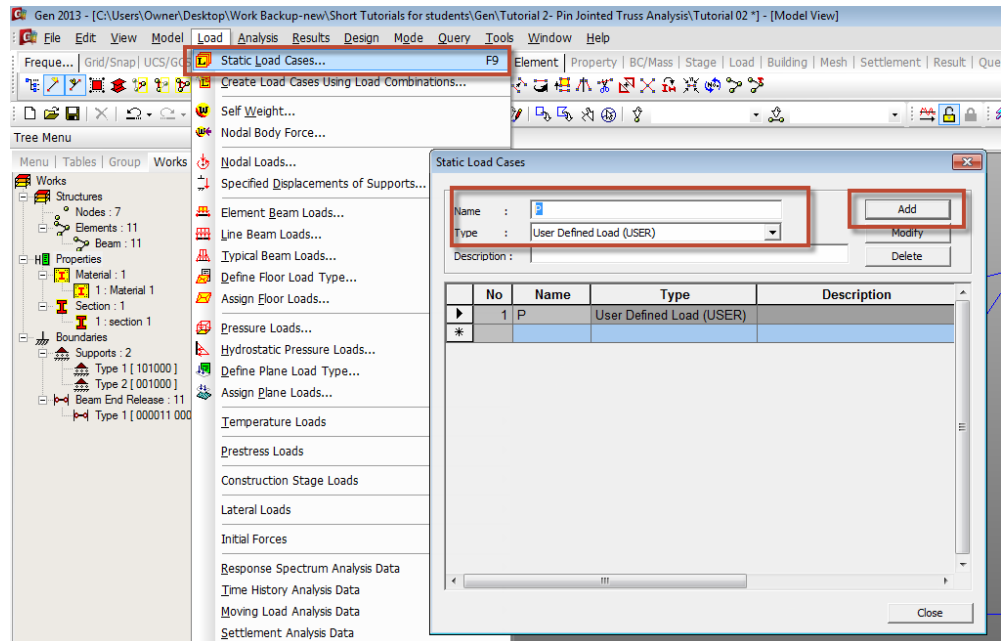
This becomes the Roller support at end C.

Pin-Jointed members: We have used beam element for modelling the members of the truss. A beam element has rotational degrees of freedom which means that beams can produce moments at the joints. Our problem description requires us to have pinned joints everywhere. To ensure that the members are all pinned, we use a function called Beam End Release. Go to **Model>Boundaries>Beam End Release**. Select all beam elements in the model by clicking the Select All button . Then select condition **Pinned-Pinned** and click Apply.

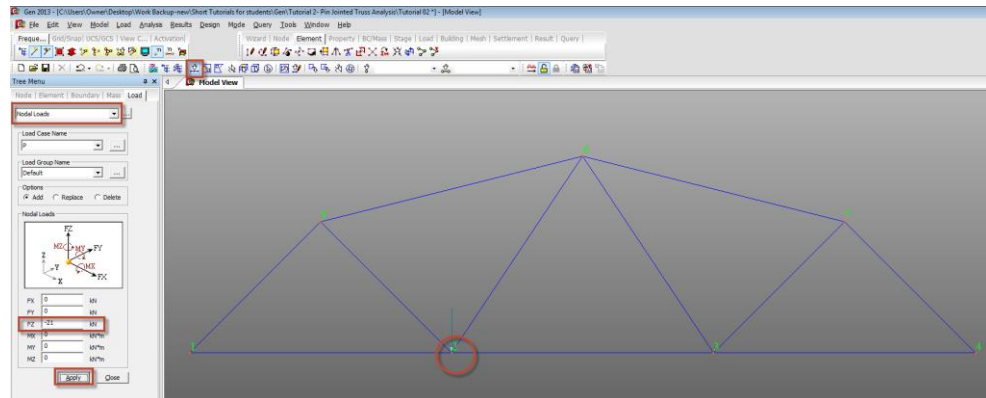


- **Load Case:** A vertically downward concentrated load, 21 kN is applied at the node 2 (Point C) in (-Z) direction and a horizontal concentrated load, 6kN is applied at node 7 (Point F) in (+X) direction.

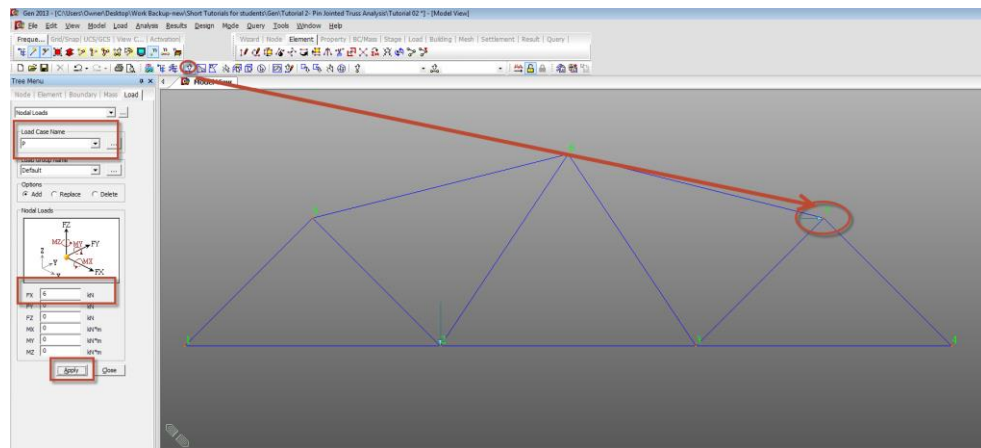
Step 11: Go to **Load>Static Load Cases** and define static load case 'P'. Select Load type as User defined for both of them. Click Close after adding the load case.



Step 12: Go to **Load>Nodal Loads** and select load case P. Select the node 2 and enter FZ=21 kN and press Apply.



Step 13: Go to **Load>Static Loads>Nodal Loads** and select load case P. Select the node 7 and enter FX=6 kN and press Apply and Close.

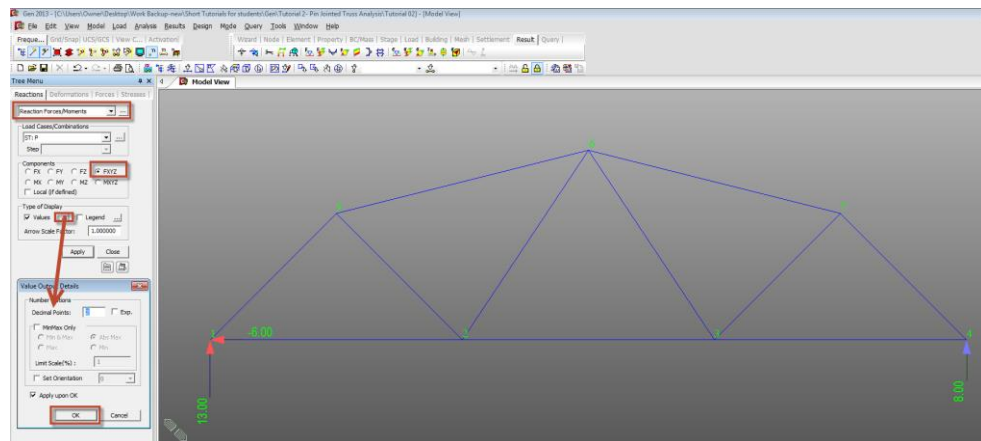


- Analysis:**
Step 13: Go to **Analysis>Perform Analysis**. After the Analysis you will find in the Message Window below showing some warnings. These warnings resulted because of the pinned joints. If all joints in the model are pinned, the structure might fail if there is any undesired external moment on the structure. However in our case, all loads are in X-Z plane and they are directly applied at the joints, thus eliminating moments. So we can safely ignore the warnings.

Results

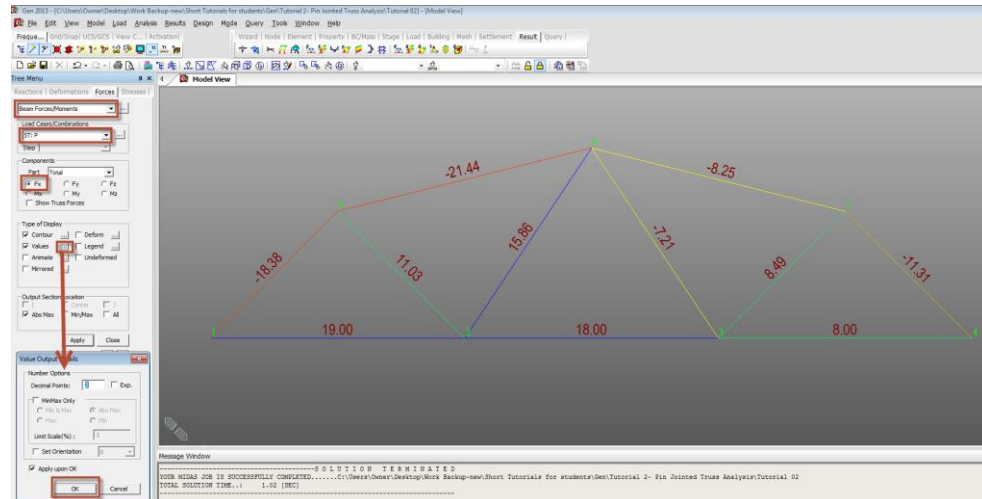
- Reaction Forces:**

Step 14: Click on **Results>Reactions>Reaction Forces/Moments** and select the load case P. Select FXYZ. Check on Values and click on the box ... next to Values to change number of decimal points to 2 and click OK to see reactions graphically.

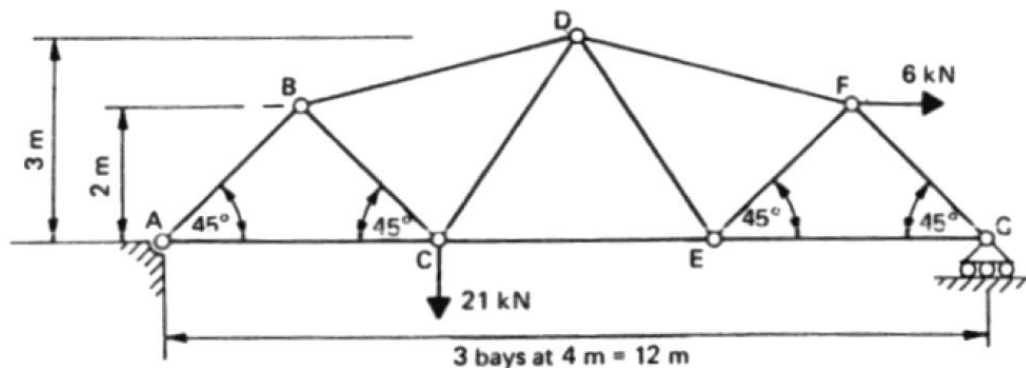


- **Member Forces:**

Step 15: Click on **Results>Forces>Beam Forces/Moments** and select the load Case P. Select Fx and Click OK to display the axial forces in the members. Note that the '+' sign indicates Tension and '-' indicates Compression.



Hand Calculations:



(1) To determine support reactions

Taking moments about A:

$$\Sigma M_A = 0$$

$$+(21 \times 4) + (6 \times 2) - (V_G \times 12) = 0$$

$$\therefore V_G = +8.00 \text{ kN}$$

$$\Sigma V = 0$$

$$+V_A + V_G - 21 = 0$$

$$V_A + (+8.00) - 21 = 0$$

$$\therefore V_A = +13.00 \text{ kN}$$

$$\Sigma H = 0$$

$$+H_A + 6 = 0$$

$$\therefore H_A = -6.00 \text{ kN}$$

Check by taking moments about G:

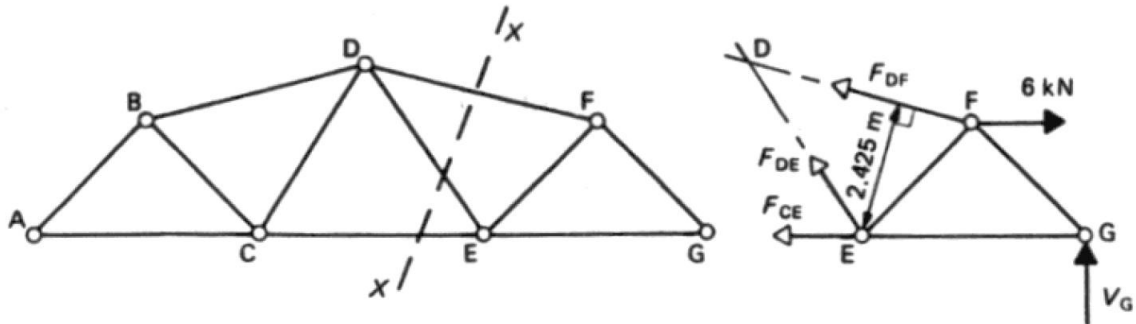
$$\Sigma M_G = 0$$

$$+(V_A \times 12) + (6 \times 2) - (21 \times 8) = +(+13.00 \times 12) + 12 - 168$$

$$= 0 \quad (\text{correct})$$

(2) To determine member forces

Cut the frame at section X-X as shown in the figure below and consider the equilibrium of the right-hand part.



Taking moments about D:

$$(\Sigma M_D = 0)$$

$$+(F_{CE} \times 3) - (6 \times 1) - (V_G \times 6) = 0$$

$$+(F_{CE} \times 3) - (6 \times 1) - (+8 \times 6) = 0$$

$$\therefore F_{CE} = +18.00 \text{ kN} \quad (\text{tensile})$$

Taking moments about E:

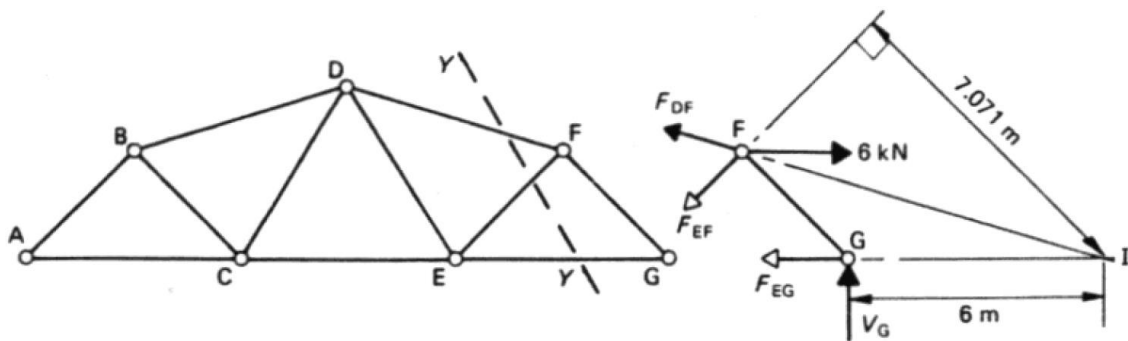
$$(\Sigma M_E = 0)$$

$$-(F_{DF} \times 2.425) - (V_G \times 4) + (6 \times 2) = 0$$

$$-(F_{DF} \times 2.425) - (+8 \times 4) + (6 \times 2) = 0$$

$$\therefore F_{DF} = -8.25 \text{ kN} \quad (\text{compressive})$$

Cut the frame at section Y-Y and consider the right-hand part as shown in figure below.



Taking Moments about I:

I is the intersection point of members DF and EG extended as shown above.

$$(\Sigma M_I = 0)$$

$$\begin{aligned}
 -(F_{EF} \times 7.071) + (6 \times 2) + (V_G \times 6) &= 0 \\
 -(F_{EF} \times 7.071) + (6 \times 2) + (+8 \times 6) &= 0 \\
 \therefore F_{EF} &= +8.48 \text{ kN} \quad (\text{tensile})
 \end{aligned}$$

The force in member GF is best determined by resolving at joint G.

Resolving at joint G:

$$(\Sigma V = 0)$$

$$\begin{aligned}
 +F_{FG} \sin 45^\circ + V_G &= 0 \\
 +F_{FG} \sin 45^\circ + (+8) &= 0 \\
 \therefore F_{FG} &= -11.31 \text{ kN} \quad (\text{compressive})
 \end{aligned}$$

Comparison of Results

Unit : kN

Reactions	Node Number	Theoretical	Midas Gen
H _A	1	-6.00	-6.00
V _A	1	13.00	13.00
V _G	4	8.00	8.00

Member forces	Element Number	Theoretical	Midas Gen	Type
FD	6	8.25	-8.25	Compression
FE	11	8.48	8.49	Tension
FG	7	11.31	-11.31	Compression
CE	2	18.00	+18.00	Tension

Reference

Ray Hulse and Jack Cain, “*Structural Mechanics, Macmillan College Workout Series*”, 1st Edition, The Macmillan Press Limited, Houndmills, Basingstoke, Hampshire, RG21 2XS, 1991, Example 2.6, Page 50.