Statically Indeterminate Frame Analysis

Title

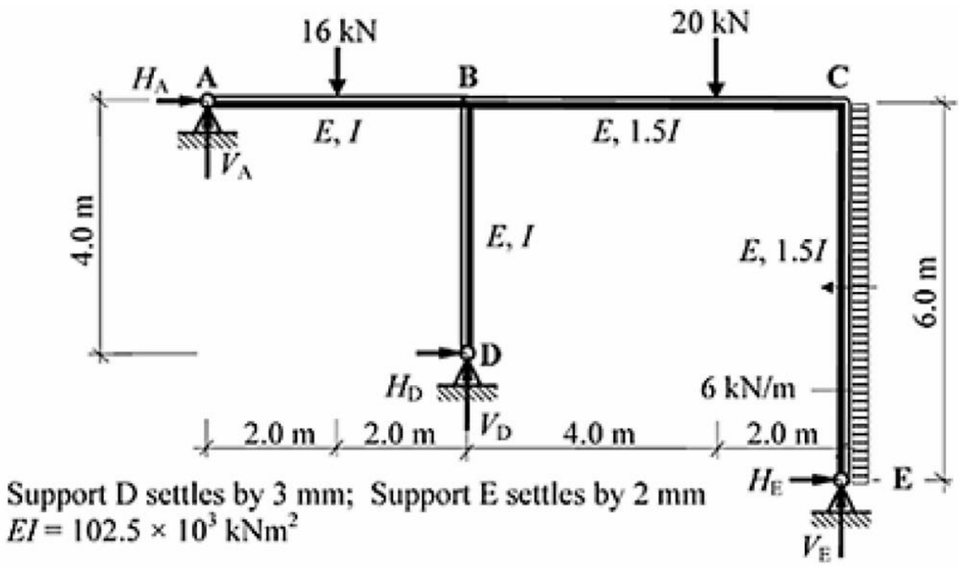
Statically indeterminate structural analysis for a No-Sway Rigid Jointed Frame

Description

A rigid-jointed, two-bay rectangular frame is pinned at supports A, D and E and carries loading as indicated in Figure below. Given that supports D and E settle by 3 mm and 2 mm respectively and that EI=102.5×10^3 kNm^2;

i) sketch the bending moment diagram and determine the support reactions,

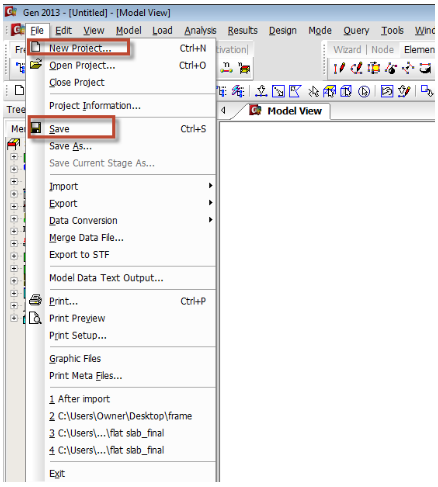
ii) sketch the deflected shape (assuming axially rigid members) and compare with the shape of the bending moment diagram (the reader should check the answer using a computer analysis solution).



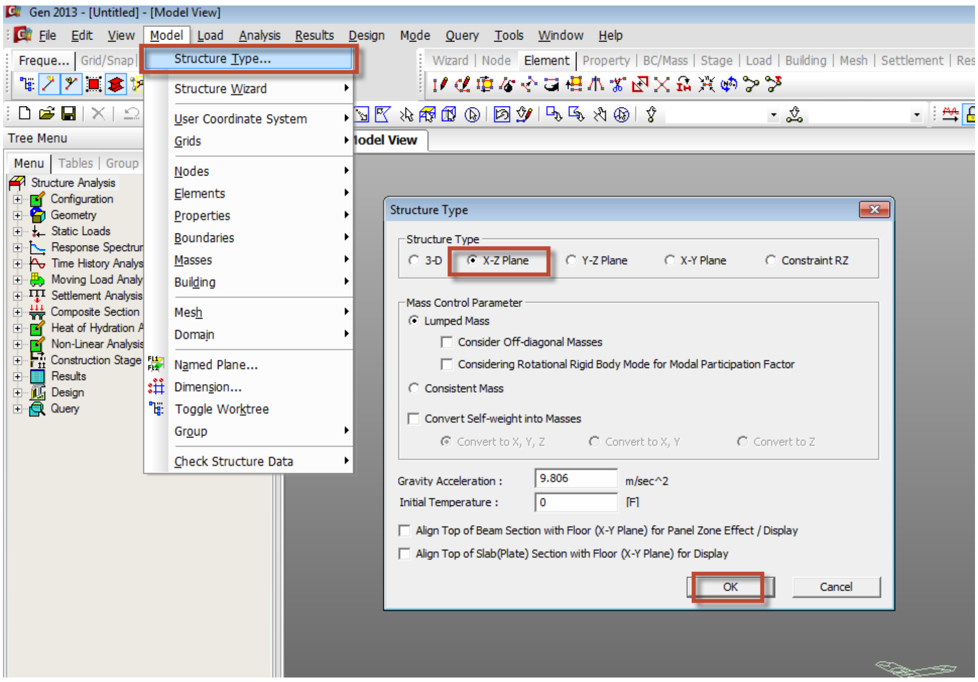
Structural geometry and analysis model

Finite Element Modelling:

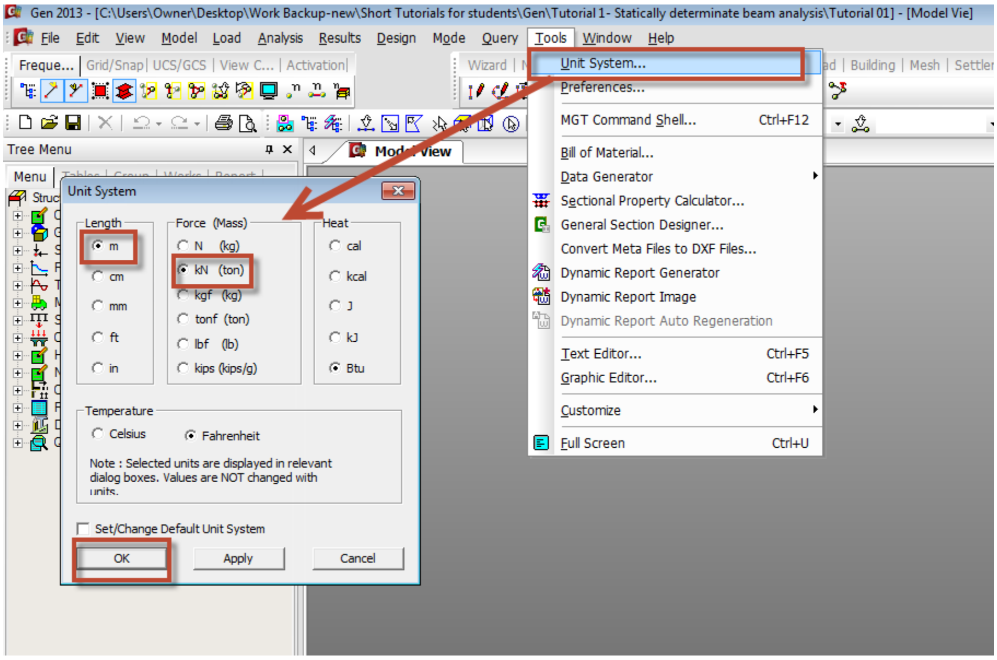
- ***Analysis Type:*** 2-D static analysis (X-Z plane)
  Step 1: Go to **File>New Project** and then go to **File>Save** to save the project with any name



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

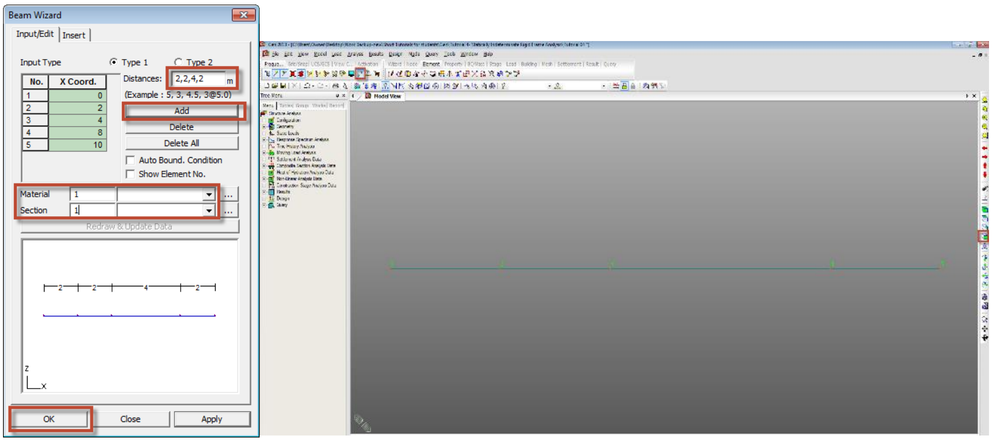


- ***Unit System:*** kN,m
  Step 3: Go to **Tools>Unit System** and change the units to kN and m.

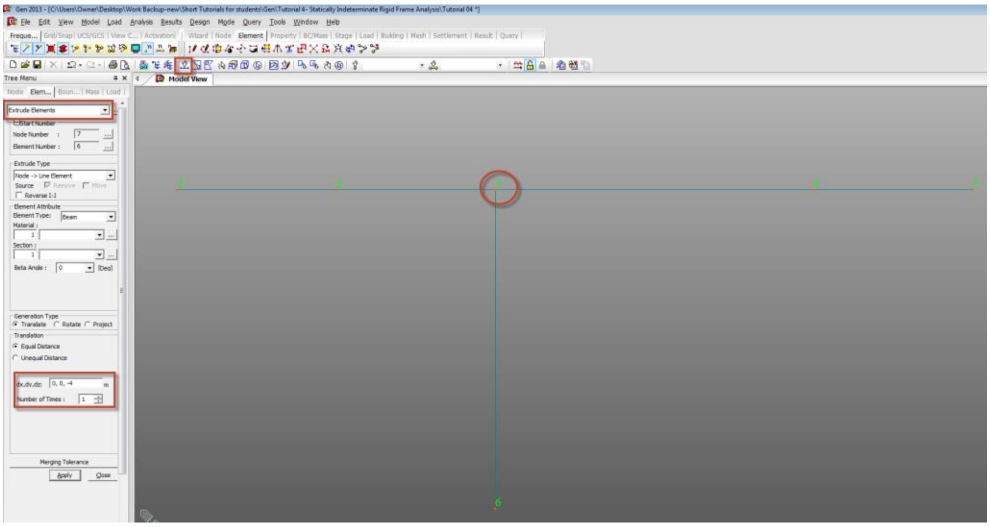


- ***Geometry generation:***

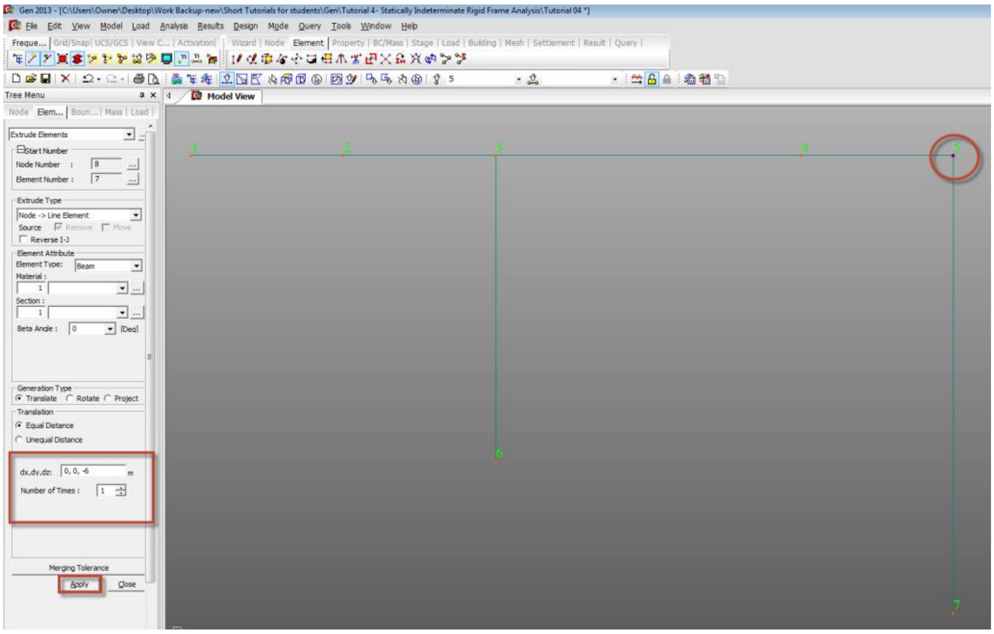
  Step 4: Go to **Model>Structure Wizard>Beam** and type 2.0,2.0,4.0,2.0 in the Distances box. Press Apply. Switch to the Front view by clicking on as shown below. Switch on node number from the option . Node number 1 will be point A.



Step 5: Go to **Model>Elements>Extrude Elements**. Select type as Node->Line. Use select single button to highlight the node number 3 by clicking on it. Type in (0,0,-4) in the Equal Distances dx,dy,dz box and enter number of times=1 and click Apply to generate the column BD.

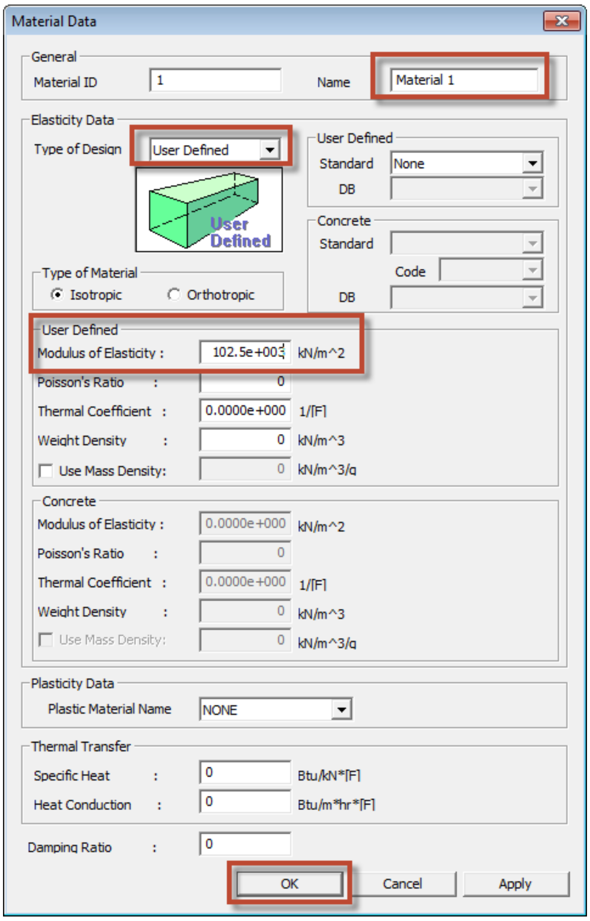


Step 6: Similar to Step 5, select this time node number 5 and enter dx,dy,dz as (0,0,-6) m, number of times=1 and click Apply to generate the second column CE.



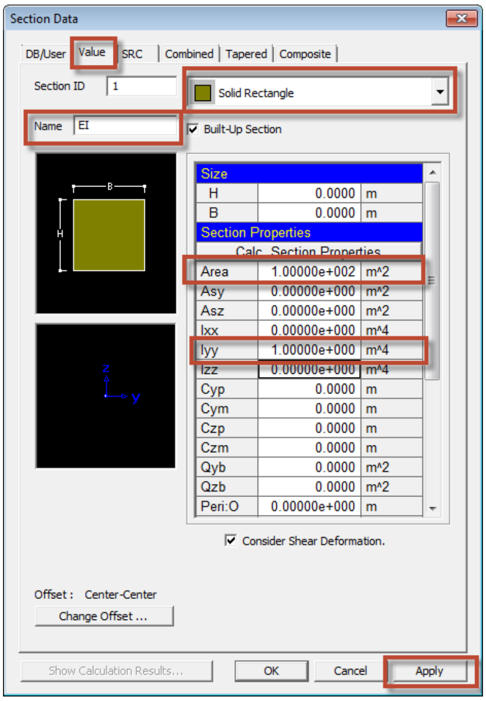
- ***Material:*** EI= 102.5*10^3 kNm^2. Let us consider Modulus of elasticity, E = 102.5 × 10^3 kN/m^2. Then I will be 1.0 m^4.

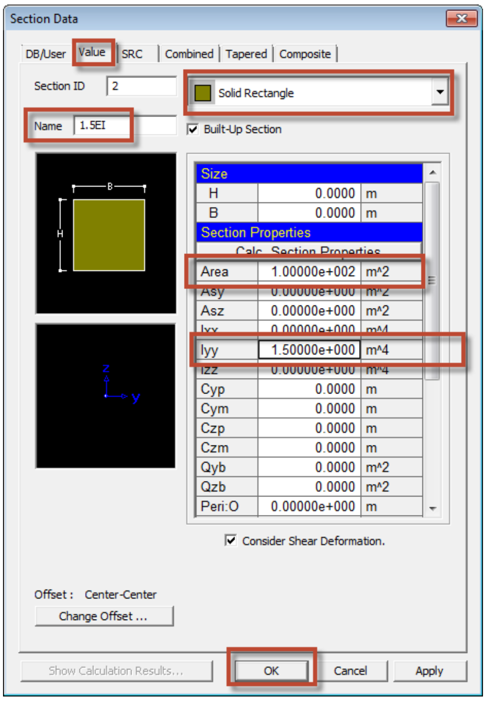
  *Step 7**:*** Go to **Model>Properties>Material>Add**. Select User defined in the Type of Design and Enter E=102.5 × 10^3 kN/m^2. Enter a name for the material and click OK and Close.



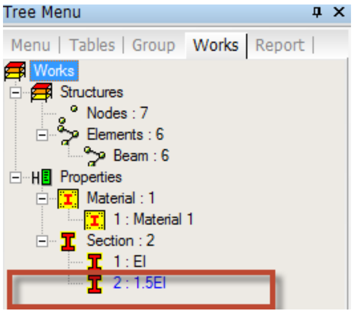
- ***Section Property:*** 2 Section properties will be defined. One for members with stiffness=EI with an Iyy value of 1.0 m^4 and a second section property for stiffness= 1.5EI with an Iyy value of 1.5 m^4

  Step 8: Go to **Model>Properties>Section>Add**. Select Value tab and select Rectangle section type. Enter Section Name as EI. Enter Area, A=100 m^4 and Iyy=1.0 m^4. Click Apply. Change Section name to 1.5EI and change Iyy to 1.5 m^4.Click OK and Close.

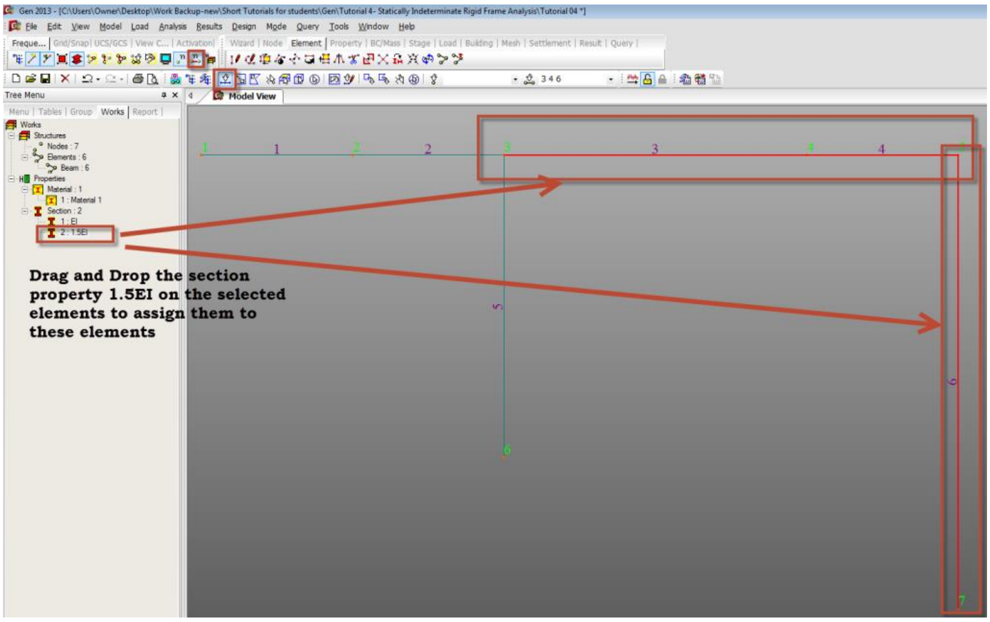




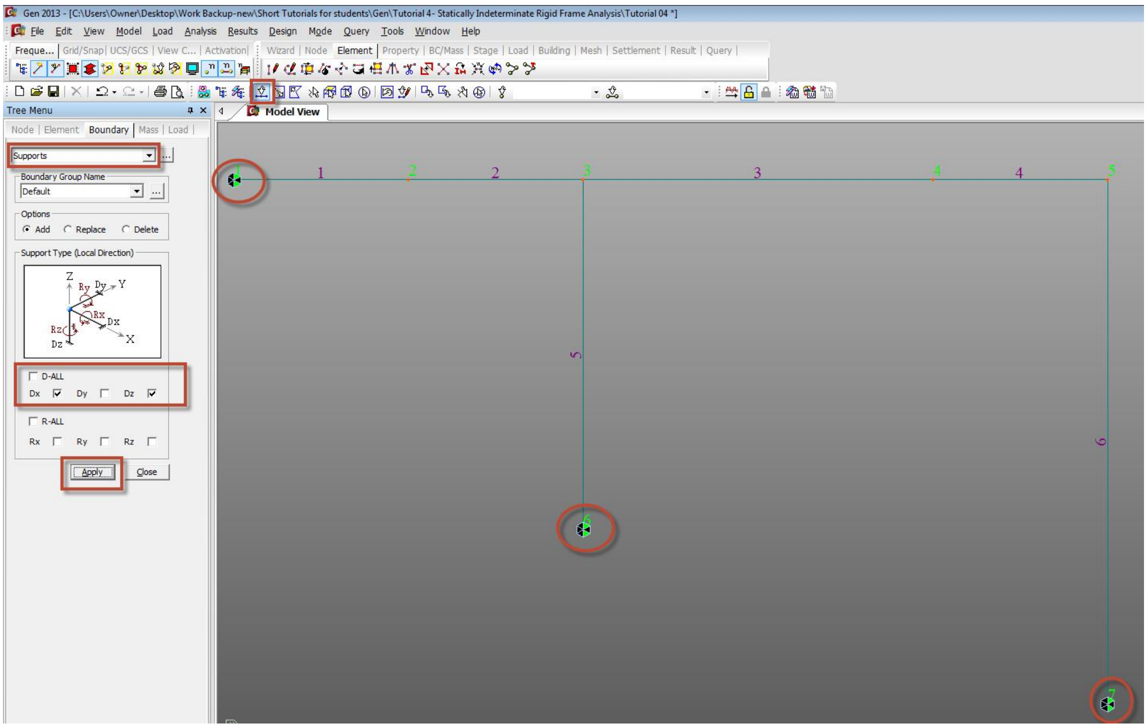
Now in the Works Tree you will see that 1 section is seen as black and 1 is seen in Blue colour. The Section number 1 is by default assigned to all elements. All assigned section properties are shown in Black whereas the sections that are yet to be assigned are always in Blue.



Display the element numbers by clicking on  and select the elements 3,4 and 6 using the select single option .



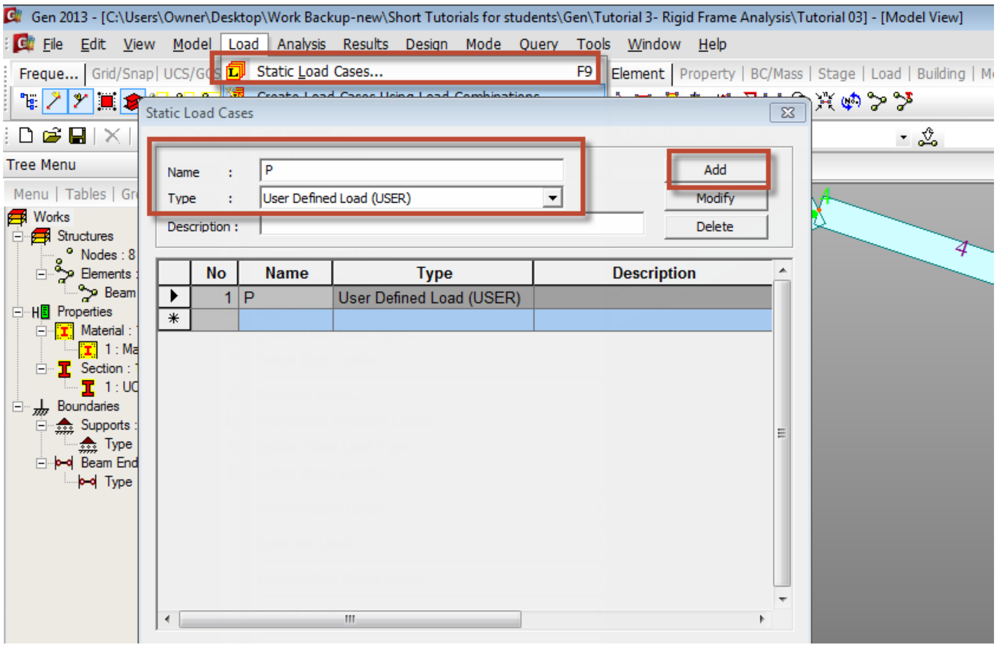
- ***Boundary Condition:*** Pinned Supports at A, D and E.
  Step 9: Use select single to select or highlight nodes 1,6 and 7 as shown in figure below. Go to **Model>Boundaries>Supports** and check on Dx and Dz and Apply.



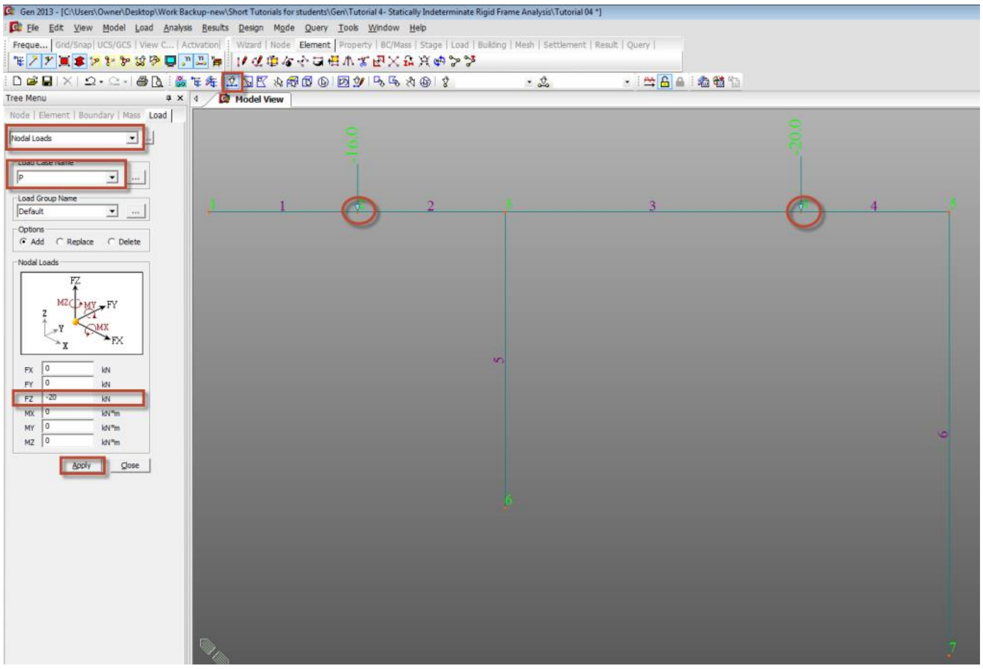
These become the Pinned supports at A,D and E. DX and DZ provide horizontal and vertical restraint simultaneously. No rotational restraint has been provided.

- ***Load Case:***
  Concentrated Loads:
  -Vertically downward loads of 16 kN and 20 kN on the members AB and BC respectively applied in (-Z) direction.
  -Horizontally applied uniformly distributed load (UDL) of 6kN/m applied on the member CE in the (-X) direction.
  -Settlements of 3mm and 2mm applied at the supports D and E respectively in the (-Z) direction.

  Step 10: 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.



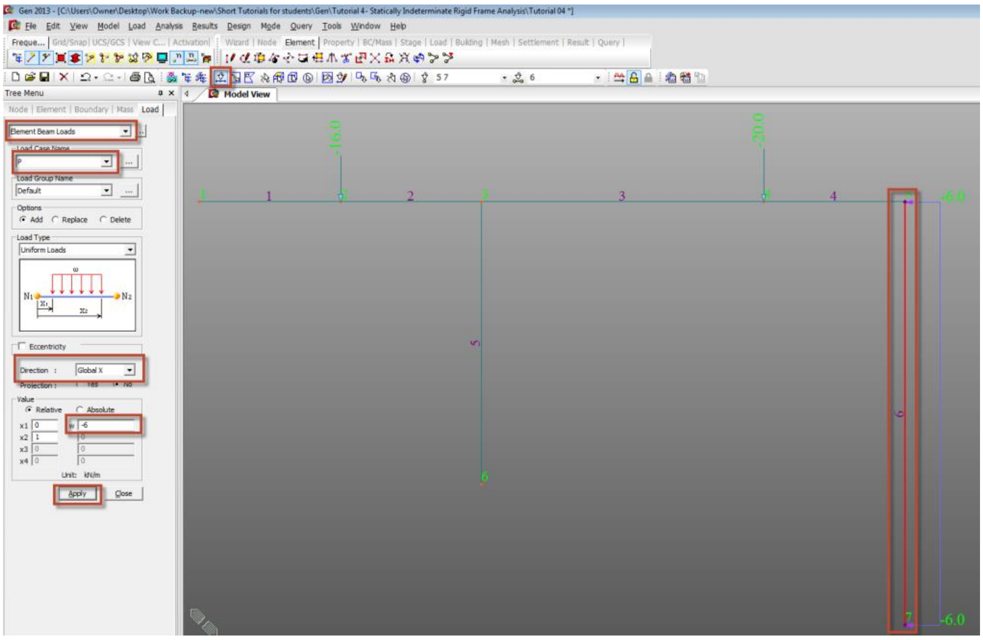
Step 11: Go to **Load>Nodal Loads** and select load case P. Select the node 2 using  and enter FZ=-16 kN and press Apply. Select Node 4 and enter FZ=-20 kN and Apply.



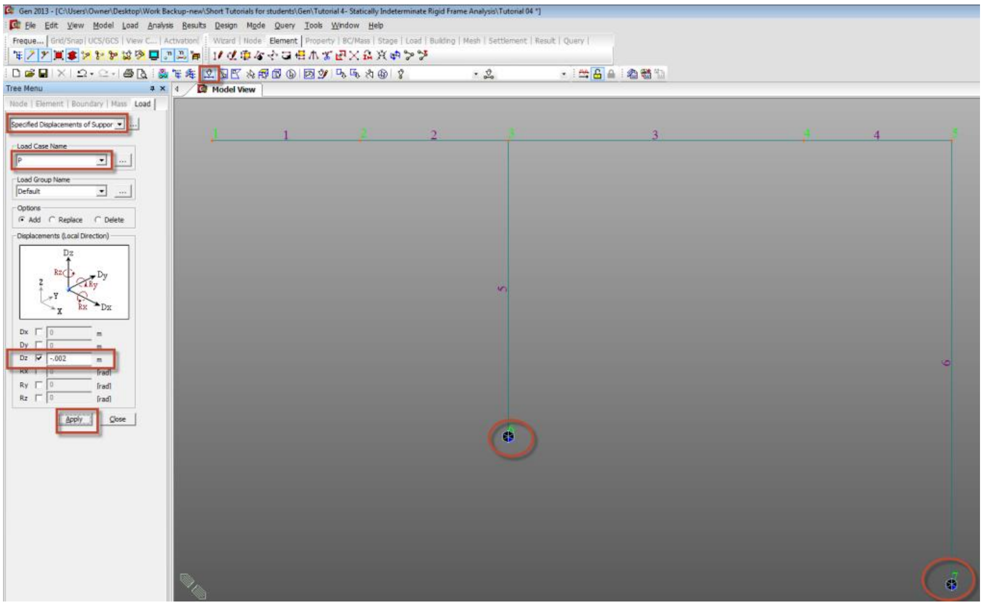
You can display the values from the Works Tree by Right clicking on Nodal Loads and click Display.

Uniform Loads: A uniformly distributed load, UDL of 12 kN/m is applied vertically downward on the member CD.

Step 12: Go to **Loads>Element Beam Loads** and select load case P. Select the element number 6 and enter w=-6 kN/m in the Global X direction and press Apply and Close.



Step 13: Go to **Loads>Specified Displacements** and select load case P. Select the node 6 and apply settlement, DZ=-3mm (-.003m). Select the node 7 and apply settlement, DZ= -.002m.



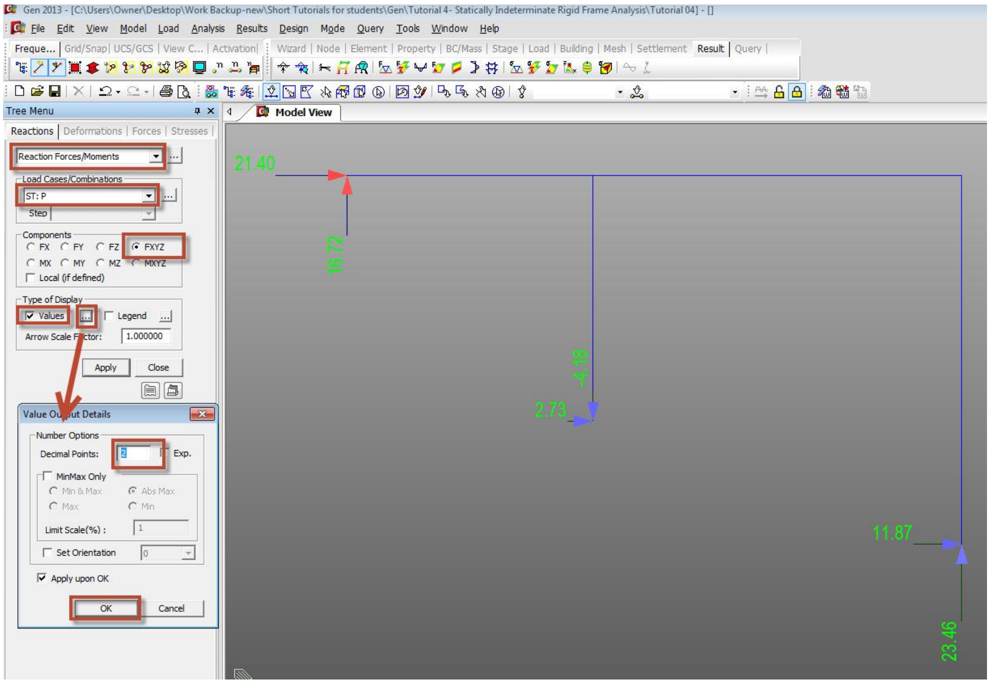
- ***Analysis:***

  Step 14: Check off node numbers and element numbers for clarity. Check off Hidden View. Go to **Analysis>Perform Analysis**

Results

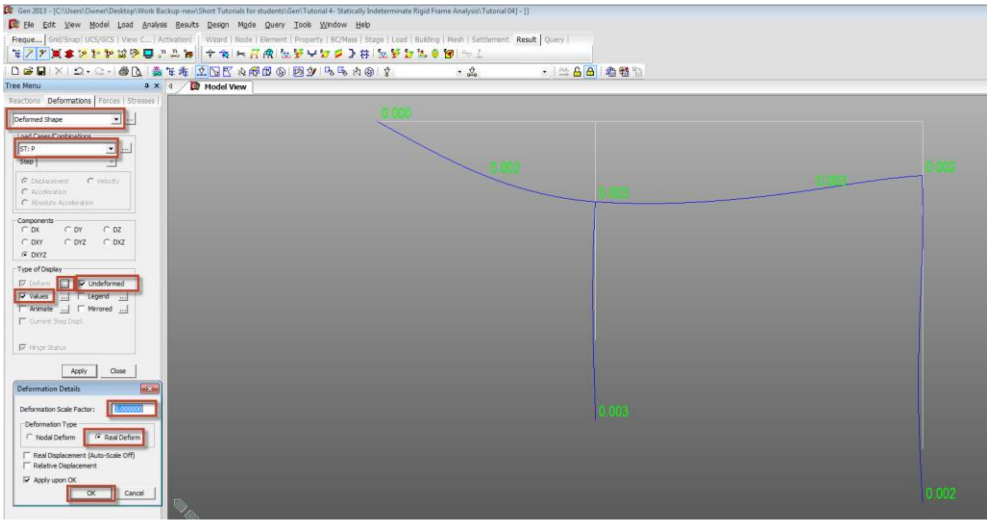
- ***Reaction Forces:***

  Step 15: 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.



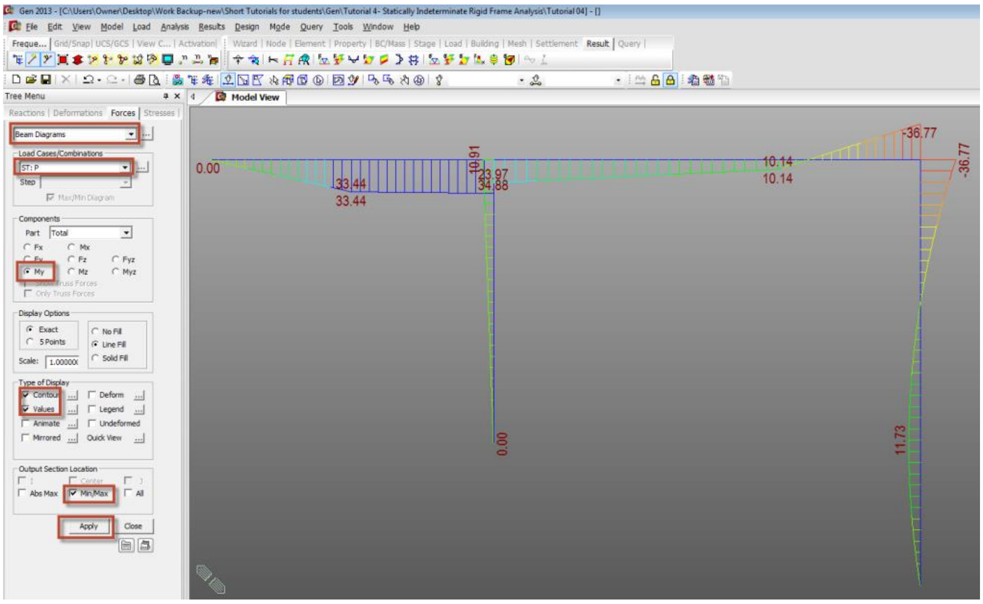
- ***Deformations:***

  Step 16: Go to **Results>Deformations>Deformed Shape**. Select load case P. Select DXYZ. Check on Values and Undeformed. Click on the box next to Deform and increase scale factor to 3 and set to real deformation and click OK.

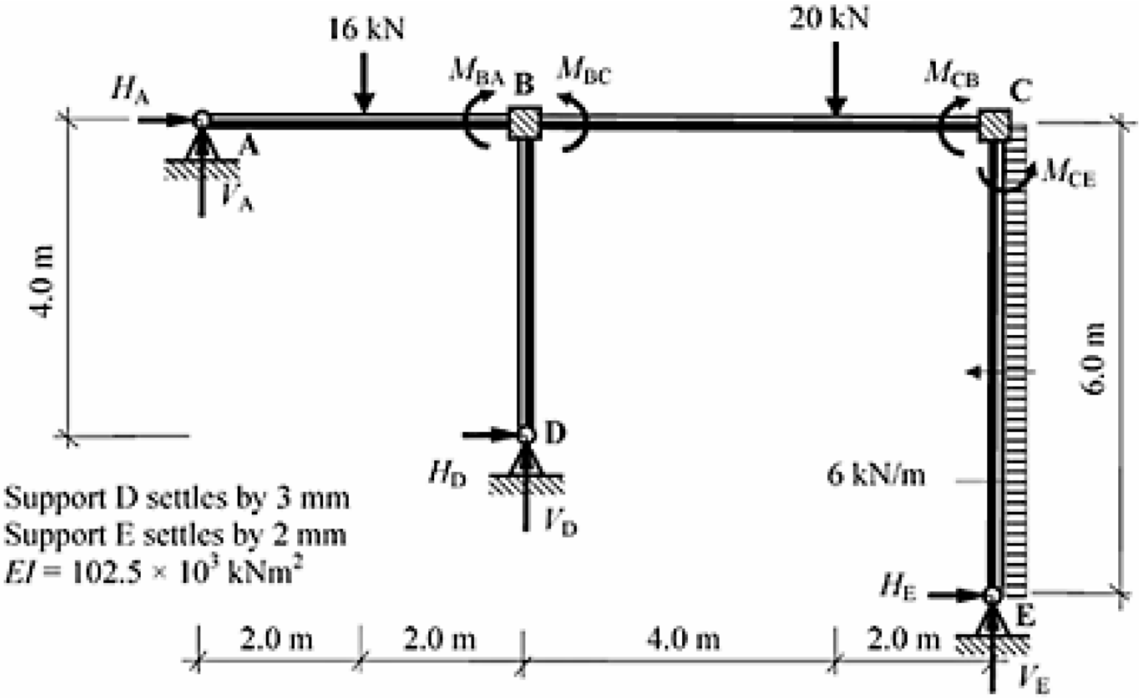


- ***Beam forces/moment diagrams:***

Step 17: Click on **Results>Forces>Beam Diagrams** and select the load Case P. Select My and Click Apply to display the bending moment diagram in the members. Select Min/Max to display the minimum and maximum bending moment values on the members.



Hand Calculations:



Moment Distribution Method will be used to calculate the bending moments.

Fixed-end Moments:

The final fixed-end moments are due to the combined effects of the applied member loads and the settlement; consider the member loads,

Member AB:

Fixed end moments

$$M_{AB} = -\frac{PL}{8} = -\frac{16\times4}{8} = -8 \text{ kNm}$$

$$M_{BA} = +\frac{PL}{8} = +\frac{16 \times 4}{8} = +8 \text{ kNm}$$

Since support A is pinned, the fixed-end moments are (M_{BA}−0.5M_{AB}) at B and zero at A.

$$(M_{BA} - M_{AB}/2) = [+8 + (0.5 \times +8)] = +12 \text{ kNm}$$

Member BC:

$$M_{BC} = -\frac{Pab^2}{L^2} = \left[-\left(\frac{20.0 \times 4.0 \times 2.0^2}{6^2}\right)\right] = -8.9\ kNm$$

$$M_{CB} = +\frac{Pa^2b}{L^2} = \left[+\left(\frac{20.0 \times 4.0^2 \times 2.0}{6^2}\right)\right] = -17.8\ kNm$$

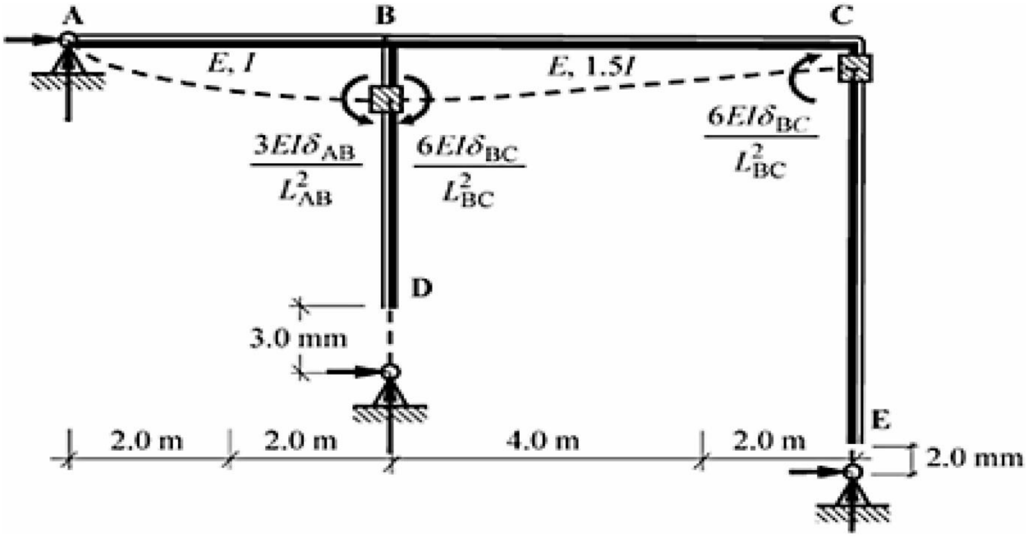
Member CE:

$$M_{CE} = -\frac{wL^2}{12} = -\frac{6.0 \times 6^2}{12} = -18.0\ kNm$$
$$M_{EC} = +\frac{wL^2}{12} = +\frac{6.0 \times 6^2}{12} = +18.0\ kNm$$

Since support E is pinned, the fixed-end moments are (MCE−0.5MEC) at C and zero at E.

$$(M_{CE} - 0.5M_{EC}) = -18.0 - (0.5 \times +18.0) = -27.0\ kNm$$

Consider the settlement of supports D and E: δ_{AB}=3.0 mm and δ_{BC}=1.0 mm



$$M_{BA} = -\frac{3(EI\delta_{AB})}{L_{AB}^2} = -\frac{3(102.5 \times 10^3 \times 0.003)}{4.0^2} = -57.6\ kNm$$

Note: the relative displacement between B and C i.e. δ_{BC}=(3.0–2.0)=1.0 mm

$$M_{BC} = +\frac{6(E1.5I\delta_{BC})}{L_{BC}^2} = +\frac{6(1.5 \times 102.5 \times 10^3 \times 0.001)}{6.0^2} = +25.6\ kNm$$

Final Fixed End Moments:

Member AB:	$M_{AB} = 0$	$M_{BA} = +12.0 - 57.6 = -45.6\ kNm$
Member BC:	$M_{BC} = -8.9 + 25.6 = +16.7\ kNm$	$M_{CB} = +17.8 + 25.6 = +43.4\ kNm$
Member CE:	$M_{CE} = -27.0\ kNm$	$M_{EC} = 0$

Distribution Factors: Joint B

$$k_{BA} = \left(\frac{3}{4} \times \frac{I}{4.0}\right) = 0.19I \qquad DF_{BA} = \frac{k_{BA}}{k_{total}} = \frac{0.19}{0.63} = 0.3$$

$$k_{BC} = \left(\frac{1.5I}{6.0}\right) = 0.25I \qquad k_{total} = 0.63I \qquad DF_{BC} = \frac{k_{BC}}{k_{total}} = \frac{0.25}{0.63} = 0.4$$

$$k_{BD} = \left(\frac{3}{4} \times \frac{I}{4.0}\right) = 0.19I \qquad DF_{BD} = \frac{k_{BD}}{k_{total}} = \frac{0.19}{0.63} = 0.3$$

Distribution Factors: Joint C

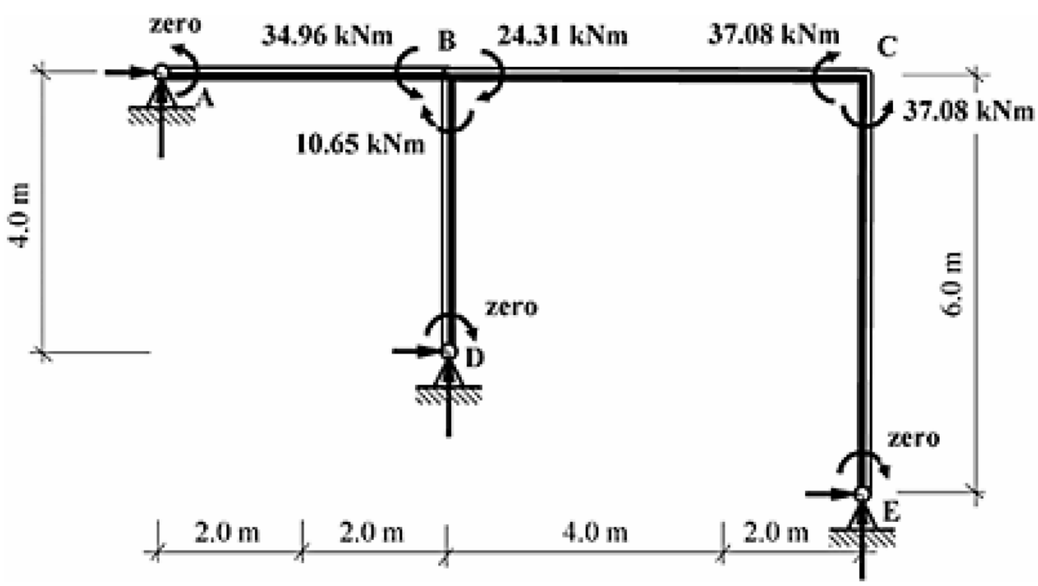
$$k_{CB} = \left(\frac{1.5I}{6.0}\right) = 0.25I \qquad k_{total} = 0.44I \qquad DF_{CB} = \frac{k_{CB}}{k_{total}} = \frac{0.25}{0.44} = 0.57$$

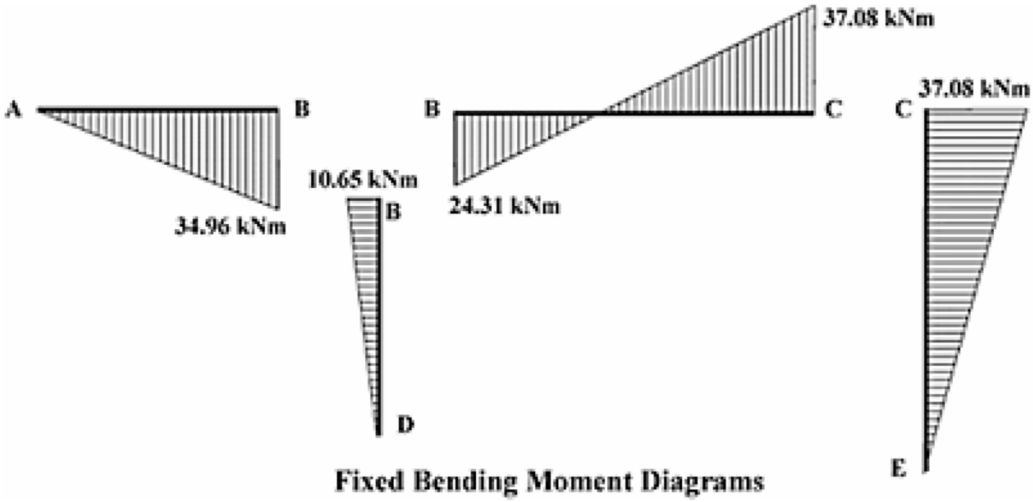
$$k_{CE} = \left(\frac{3}{4} \times \frac{1.5I}{4.0}\right) = 0.19I \qquad DF_{CE} = \frac{k_{CE}}{k_{total}} = \frac{0.19}{0.44} = 0.43$$

Moment Distribution Table:

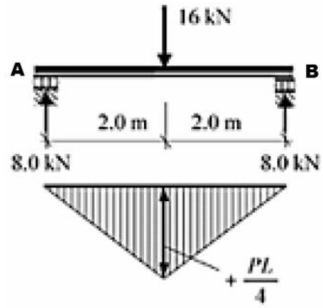
Joint	A	D		B				C			E
	AB	DB		BA	BD	BC		CB	CE		EC
Distribution Factors	1.0	1.0		0.3	0.3	0.4		0.57	0.43		1.0
Fixed End Moments				-45.60		+16.70		+43.4	-27.0		
Balance				+8.67	+8.67	+11.56		-9.35	-7.05		
Carry-over						-4.67		+5.78			
Balance				+1.40	+1.40	+1.87		-3.29	-2.49		
Carry-over						-1.65		0.93			
Balance				+0.49	+0.49	+0.66		-0.53	-0.4		
Carry-over						-0.27		+0.33			
Balance				0.08	+.08	+0.11		-0.19	-0.14		
Total	**0**	**0**		**-34.96**	**+10.65**	**+24.31**		**+37.08**	**-37.08**		**0**

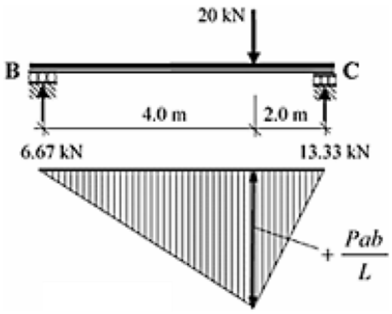
Continuity Moments:





Free Bending Moments:





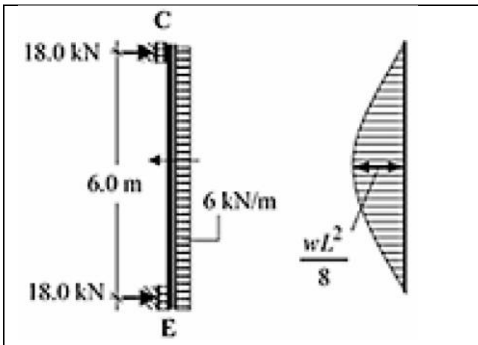
Member AB:

$$M_{free} = \frac{PL}{4} = \frac{16 \times 4}{4} = 16\ kNm$$

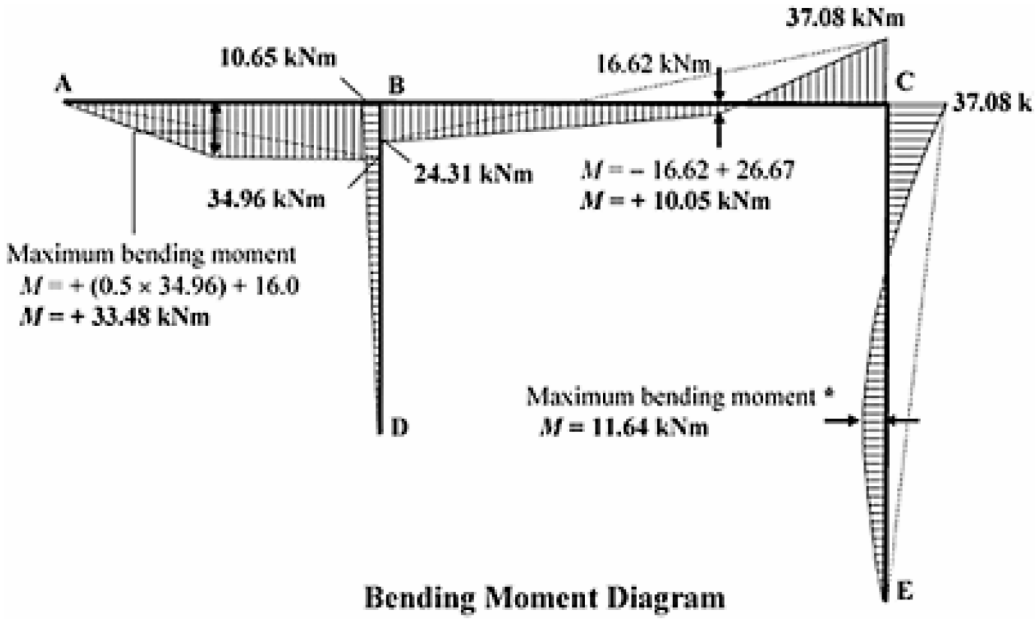
Member BC:

$$M_{free} = \frac{Pab}{L} = \frac{20 \times 4 \times 2}{6} = 26.67\ kNm$$

Member CE:

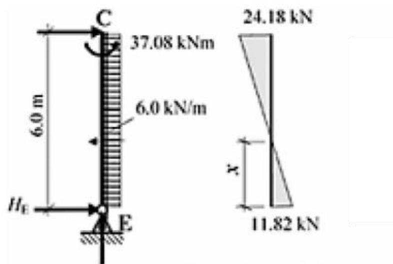


$$M_{free} = \frac{wL^2}{8} = \frac{6 \times 6.0^2}{8} = 27\ kNm$$



Bending Moment Diagram

The maximum value along the length of member CE can be found by identifying the point of zero shear as follows:



Shear Force Diagram

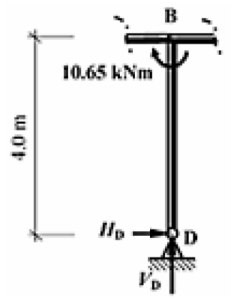
$\Sigma M_C = 0$

$+(6.0 \times 6.0 \times 3.0) - 37.08 - (H_E \times 6) = 0$

$\boldsymbol{H_E = +11.82\ kN}$

$x = (11.82/6.0) = 1.97\ m$

$M_{maximum} = (0.5 \times 1.97 \times 11.82) = 11.64\ kNm$

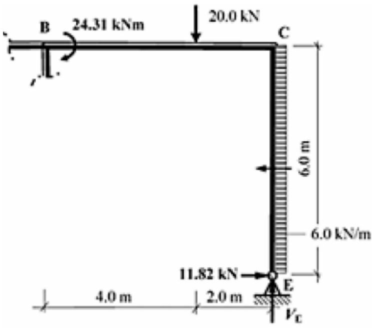


Consider Member BD

$\Sigma M_B = 0$

$+10.65 - (H_D \times 4.0) = 0$

$\therefore \boldsymbol{H_D = 2.66\ kN}$

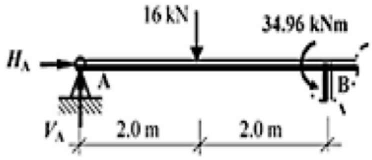


Consider a section B:

$\Sigma M_B = 0$

$$+24.31 + (20.0 \times 4.0) - (11.82 \times 6.0) + (6.0 \times 6.0 \times 3.0) - (V_E \times 6.0) = 0$$

$\therefore \boldsymbol{V_E = +23.57\ kN}$



Consider Member AB:

$\Sigma M_B = 0$

$-34.96 - (16.0 \times 2.0) + (V_A \times 4.0) = 0$

$\therefore \boldsymbol{V_A = +16.74\ kN}$

For the Complete Frame:

$\Sigma V = 0$

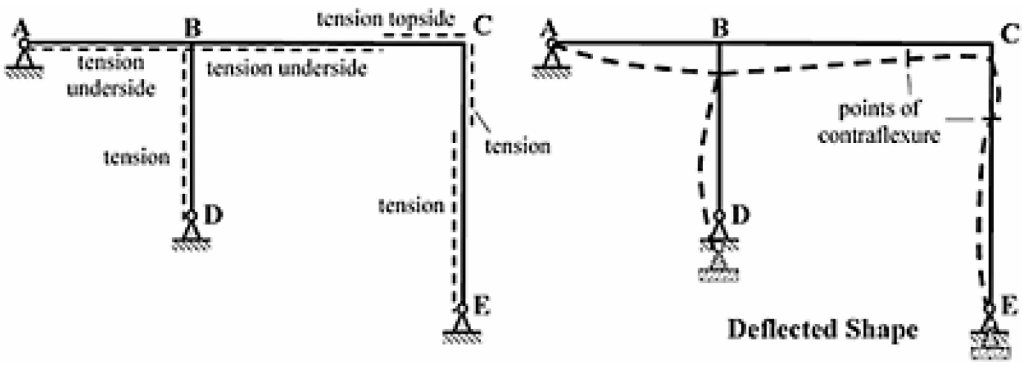
$$+16.74 - 16.0 - 20.0 + 23.57 + V_D = 0$$

$$\therefore \boldsymbol{V_D = -4.31\ kN}$$

$\Sigma H = 0$

$$H_A + 11.82 + 2.66 - (6.0 \times 6.0) = 0$$

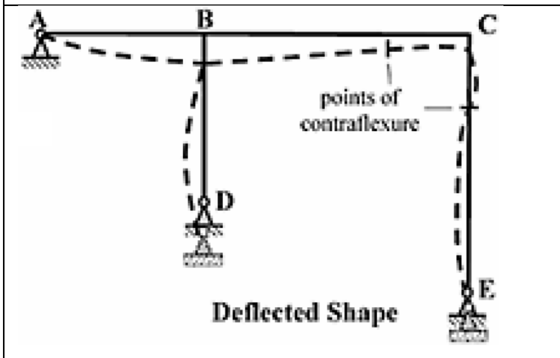
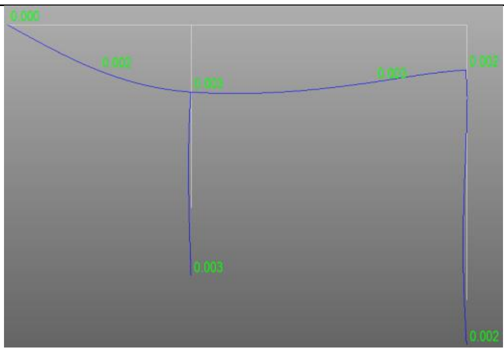
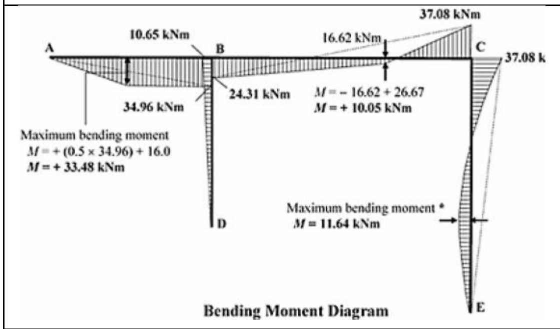
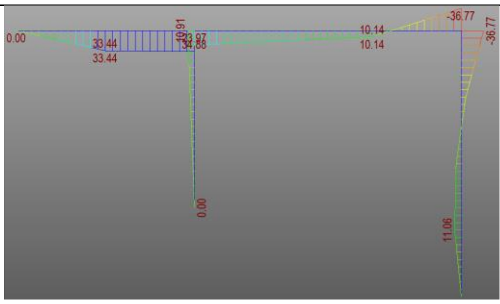
$$\therefore \boldsymbol{H_A = +21.52\ kN}$$



Comparison of Results

Unit : kN,m

Reactions	Node Number	Theoretical	Midas Gen
H_A	1	+21.52	+21.40
V_A	1	+16.74	+16.72
H_D	6	+2.66	+2.73
V_D	6	-4.31	-4.18
H_E	7	+11.82	+11.87
V_E	7	+23.57	+23.46

Theoretical Diagrams	Midas Gen Diagrams
	
	

Reference

William M.C. McKenzie, "*Examples in Structural Analysis*", 1st Edition, Taylor & Francis 2 Park Square, Milton Park, Abingdon, Oxon OX14 4RN, 2006, 5.2.1 Example 5.3, Page 383.