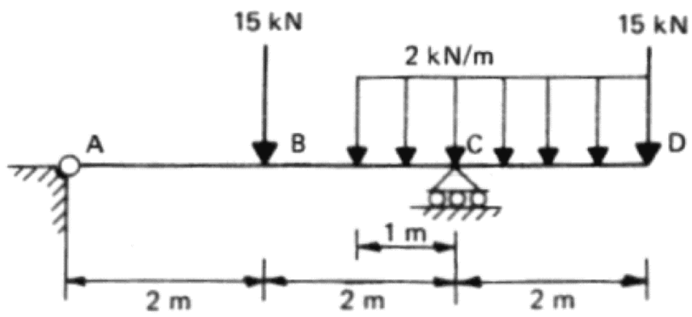
Statically determinate beam analysis

Title

Statically determinate structural analysis for a simply supported beam with an overhang for reaction calculations.

Description

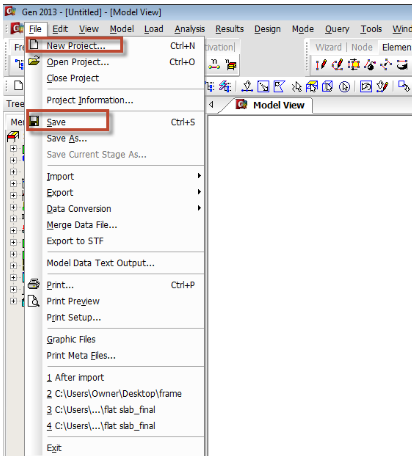
Beam ABCD has a pinned support at A and a roller support at C. It carries two concentrated loads of 15 kN each and a uniformly distributed load of 2 kN/m over the right hand half as shown in the figure below. Determine the Reactions.



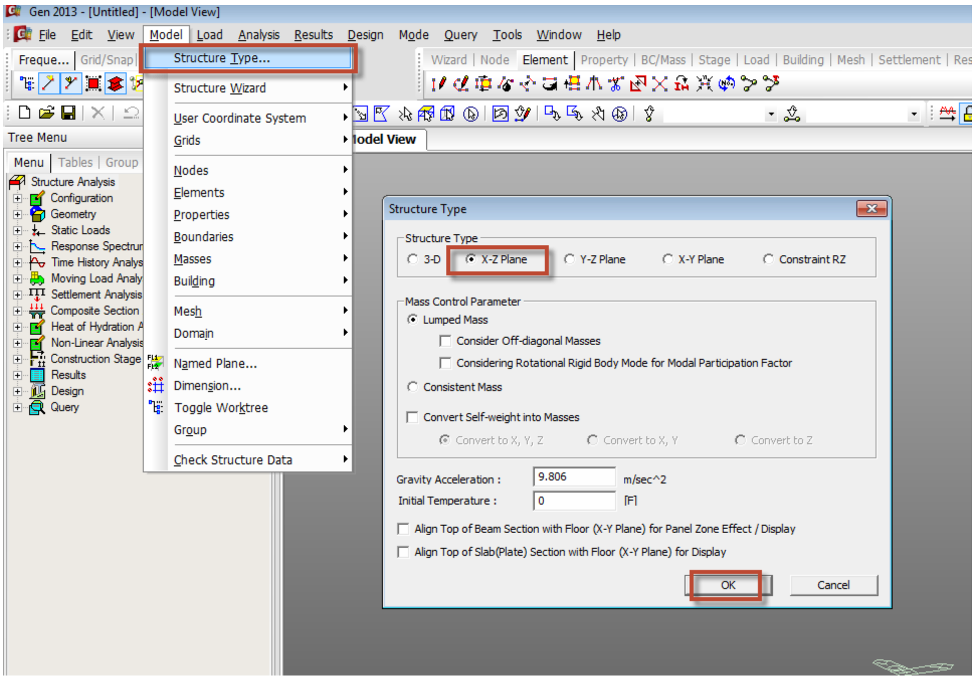
Structural geometry and analysis model

Finite Element Modelling:

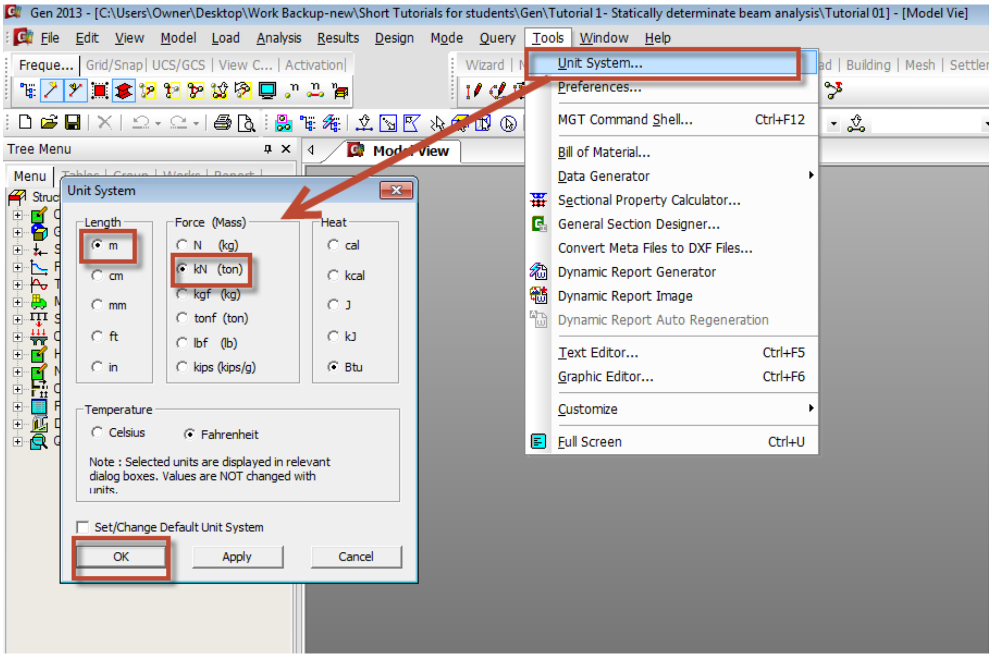
- ***Analysis Type:*** 2-D static analysis (X-Z plane)
  Step 1: Go to **File>New Project** and then go to **File>Save** to save the project with any name



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

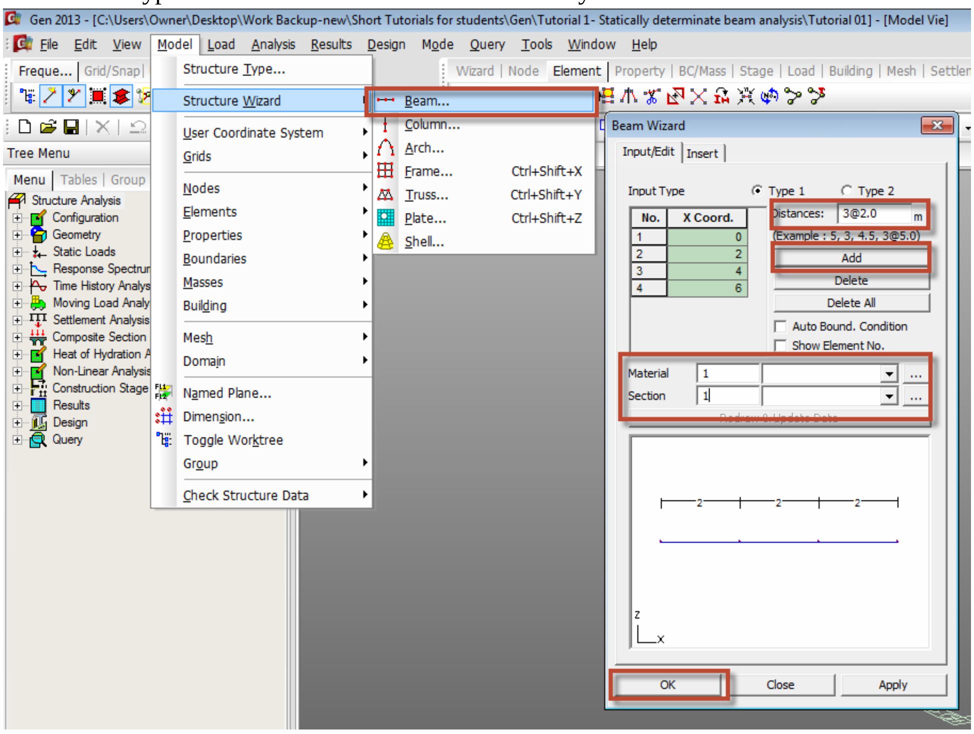


- ***Unit System:*** kN,m
  Step 3: Go to **Tools>Unit System** and change the units to kN and m. You can also change units any time in the model from the status bar below as shown in the figure.



- ***Dimension:*** Length= 3@2.0m= 6.0m

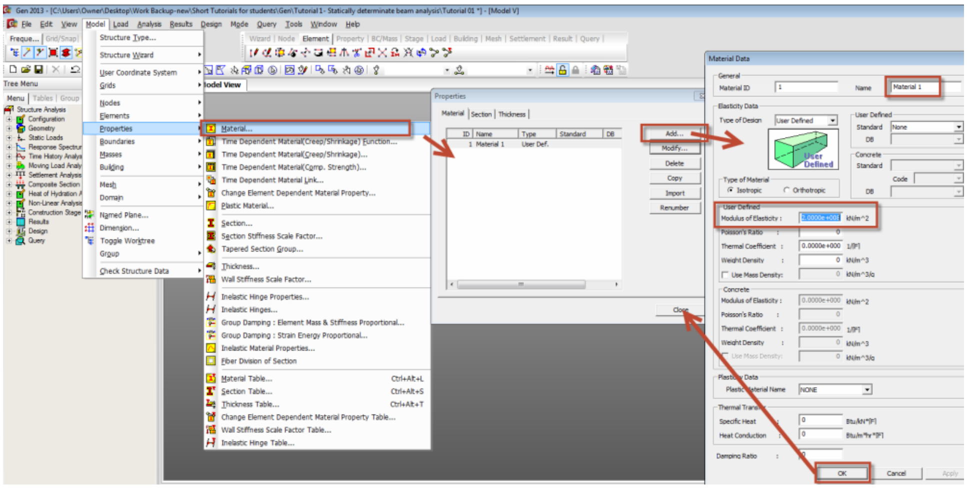
  Step 4: Go to **Model>Structure Wizard>Beam** and type in 3@2.0 in the Distances box. Press Add. Type 1 in the Material and Section ID entry.



- ***Element type:*** Beam element (Beam Wizard from Step 4 generates beam elements automatically)

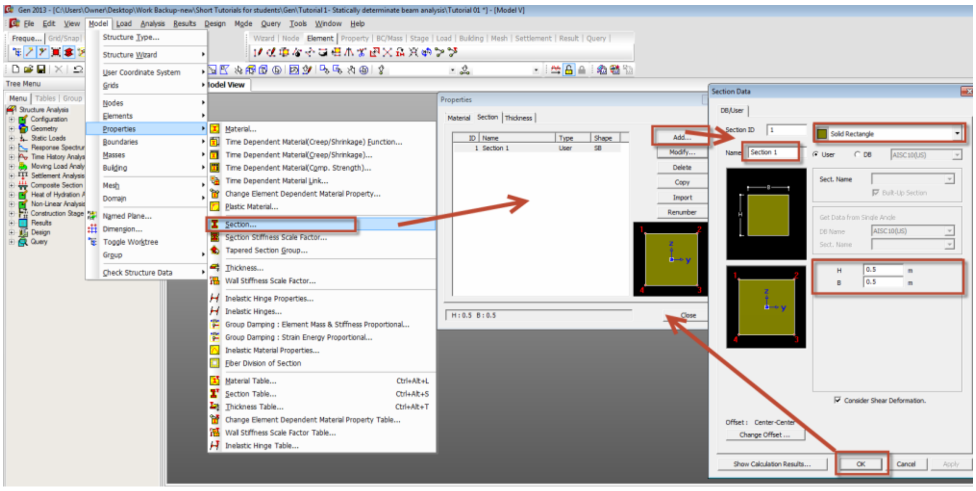
- ***Material:*** Modulus of elasticity, E = 3.0 × 10^7 kN/m^2

Step 5: Go to **Model**>**Properties**>**Material**>**Add**. Select User defined in the Type of Design and Enter E=3.0 × 10^7 kN/m^2. Enter a name for the material and click OK and Close

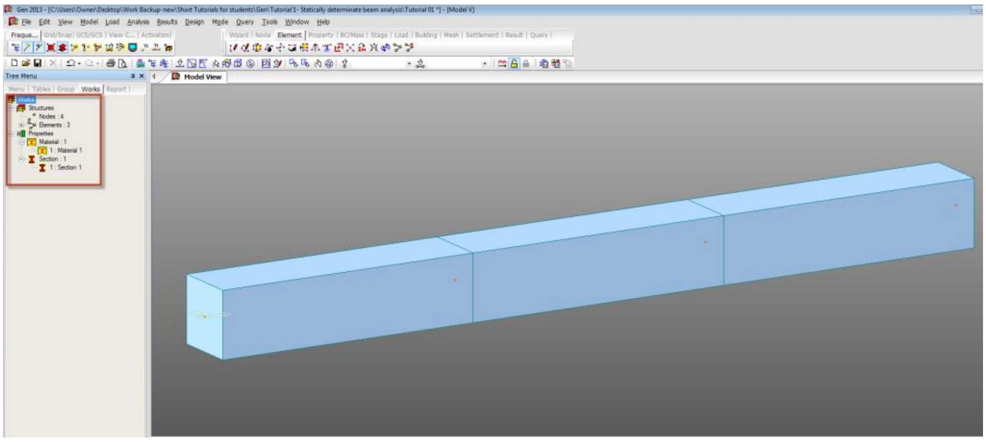


- ***Section Property:*** B × H = 0.5 m × 0.5 m

Step 6: Go to **Model**>**Properties**>**Section**>**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, click OK and Close

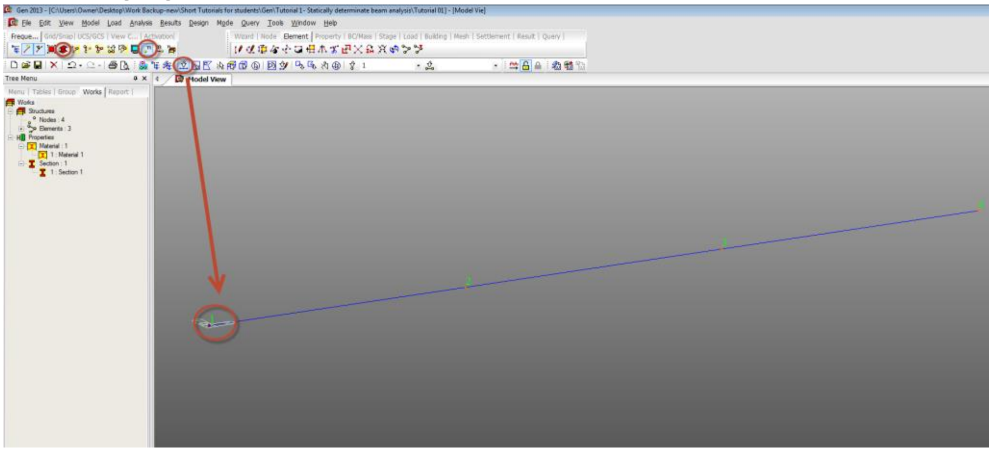


You can see the shape of the section in the model generated. Go to the Works Tree to check the information for your model.

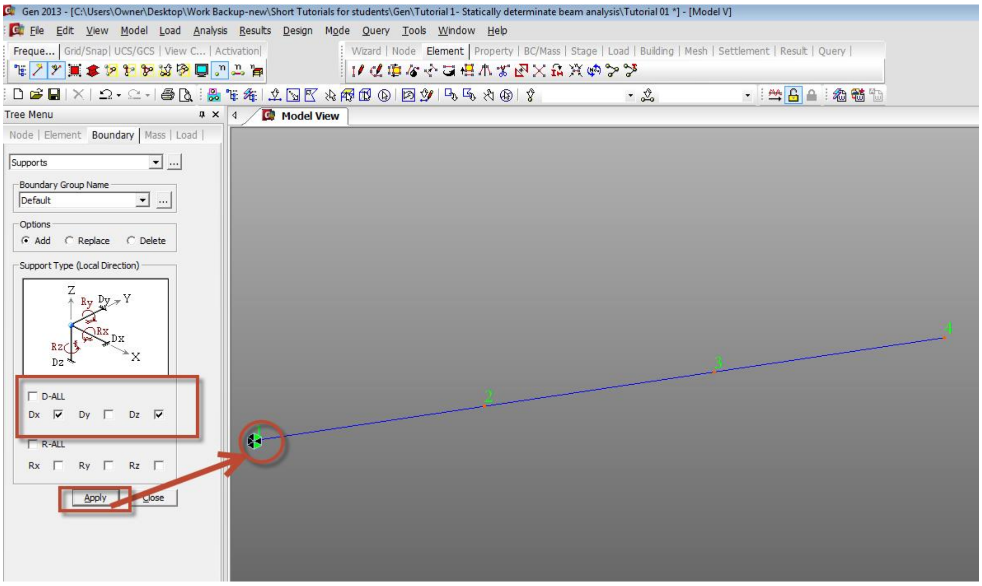


- ***Boundary Condition:*** Simply Supported (Pinned at A and Roller at C)

Step 7: Go to **View>Remove Hidden Lines or click** to toggle back to the wireframe (line) view. Check on in the toolbar to display node numbers. Use select single to select or highlight node 1.

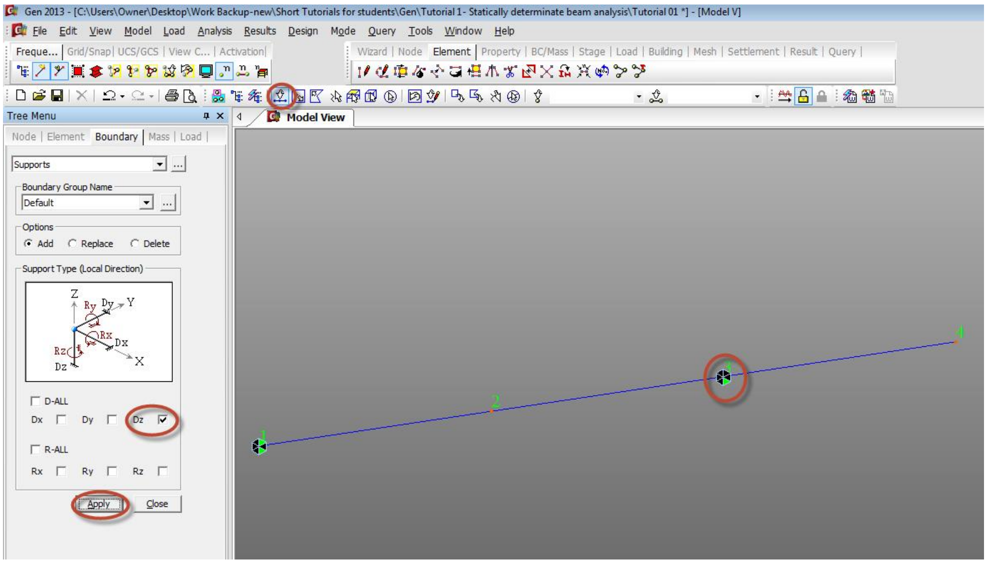


Step 7: Go to **Model>Boundary>Supports** and check on Dx and Dz and Apply.



This becomes the Pinned support.

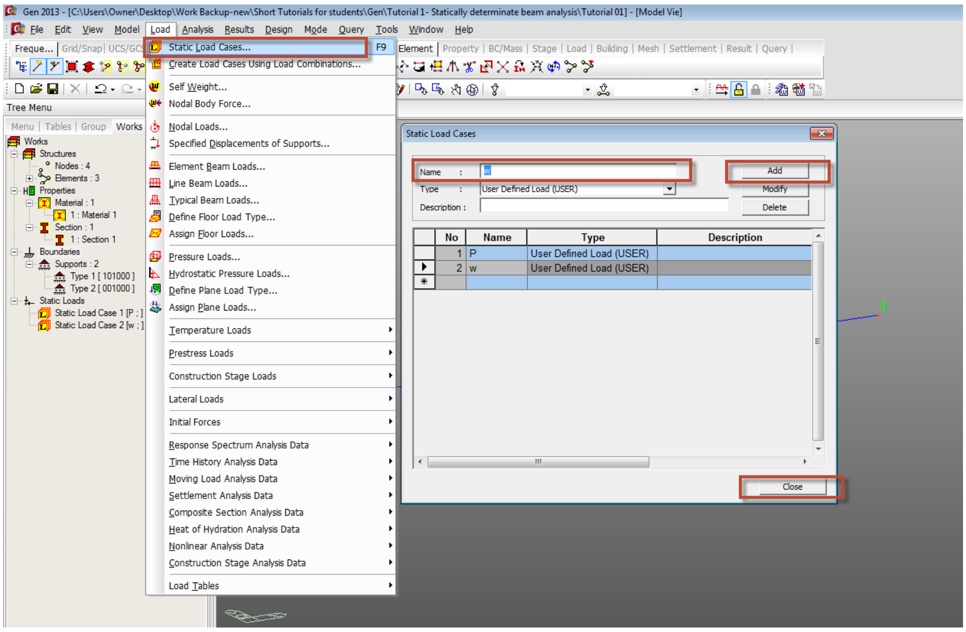
Step 8: Again use Select Single  , this time to highlight or select node 3 and check on Dz only, Apply and Close.



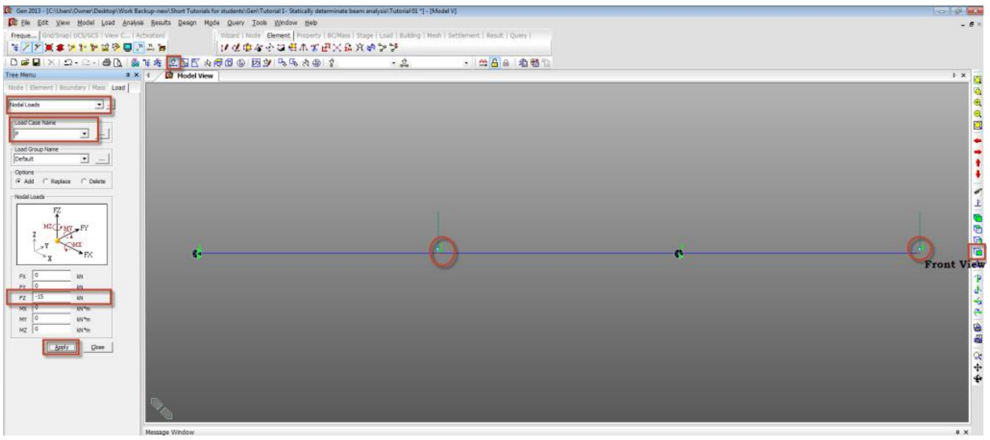
This becomes the Roller support.

- ***Load Case 1:*** 2 vertically downward concentrated loads P=15 kN are applied at the nodes 2 and 4 in the (-) Z direction.

Step 9: Go to **Loads>Static Load Cases** and define static load cases 'P' (concentrated load) and 'w' (Uniformly Distributed Load). Select Load type as User defined for both of them. Click Close after adding the two load cases.

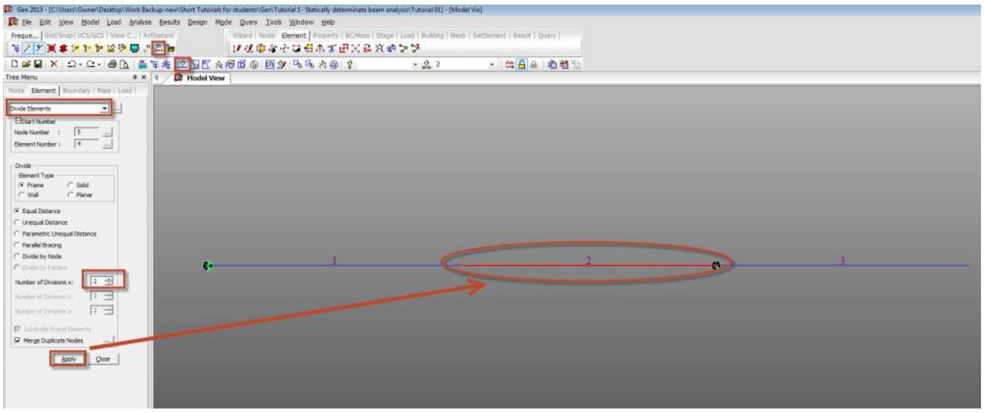


Step 10: Go to **Load>Nodal Loads** and select load case P. Select the nodes 2 and 4. Enter FZ=-15 kN and press Apply. Click on the Front View icon on the Right to see the front view of the beam.

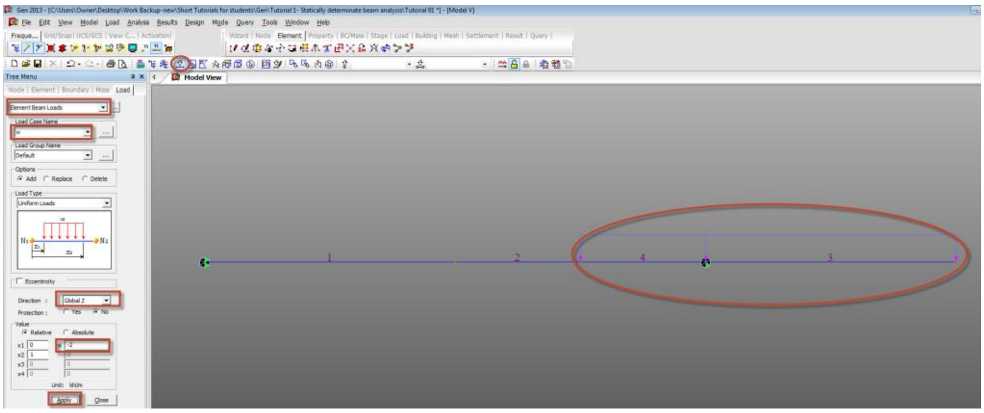


- ***Load Case 2:*** A uniformly distributed load, w=2kN/m is applied on the beam over a distance of 3m from end D.

Step 11: Switch off display of node numbers by clicking on . Click on to display element numbers. Go to **Model>Elements>Divide Elements.** Select the element number 2 using . Enter number of divisions as 2 and click Apply and Close.



Step 12: Go to **Load>Element Beam Loads**. Select the load case 'w' and select Load type as Uniform Load. Enter w=-2kN/m in the Direction Global Z. Select elements 4 and 3 using . Click Apply and Close. Switch off element number display .

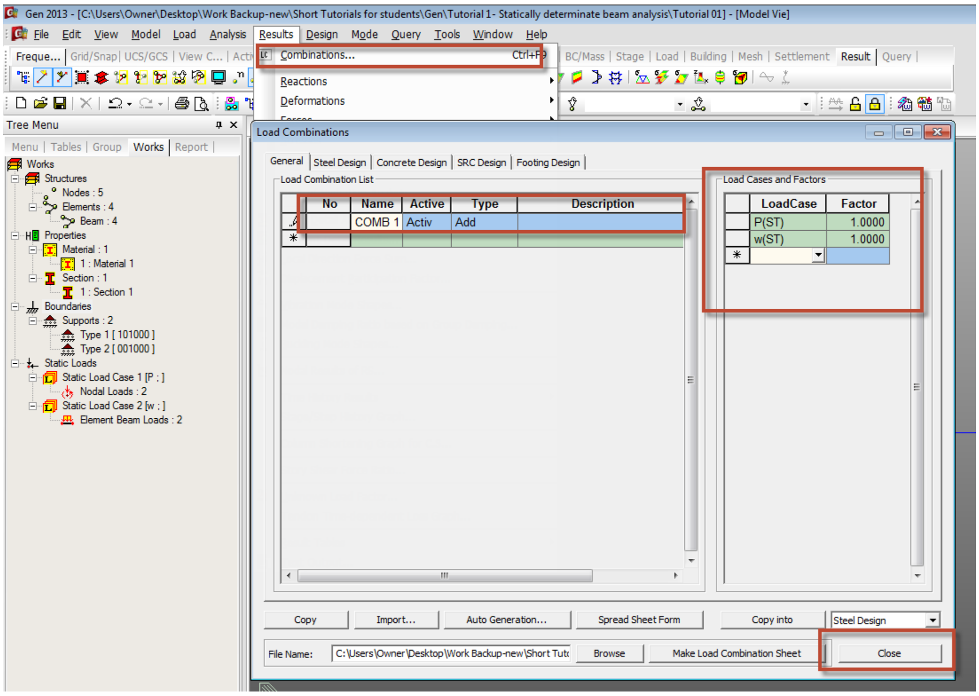


- ***Analysis:*** *Step 13:*Go to **Analysis>Perform Analysis** or Press **F5**

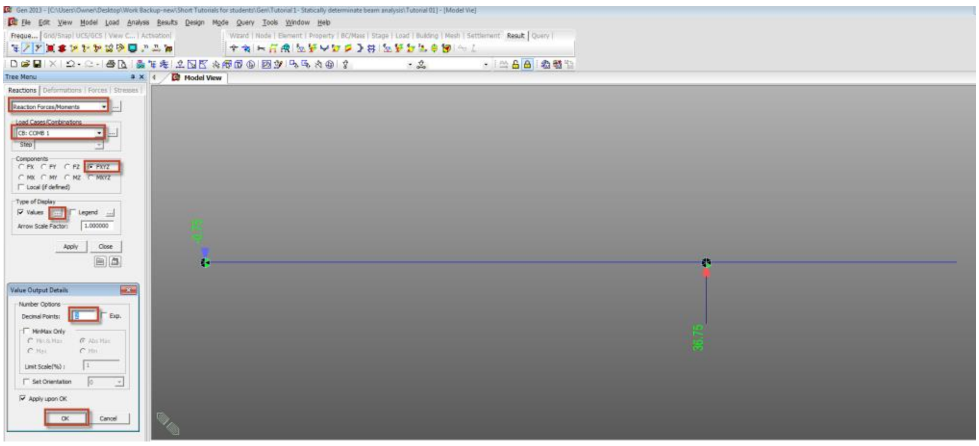
Results

Reaction Forces:

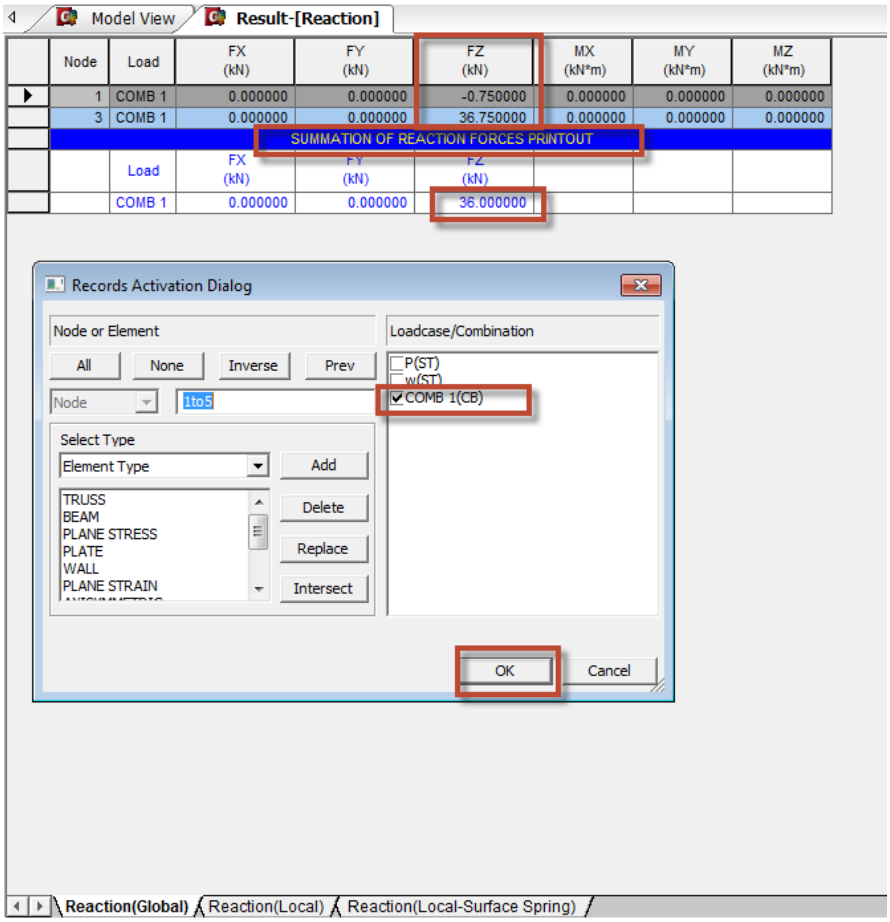
Step 14: Go to **Results>Combinations** and enter a combination name. Select type as Add and add a combination COMB 1 = 1.0*P + 1.0*w as shown in the figure below.



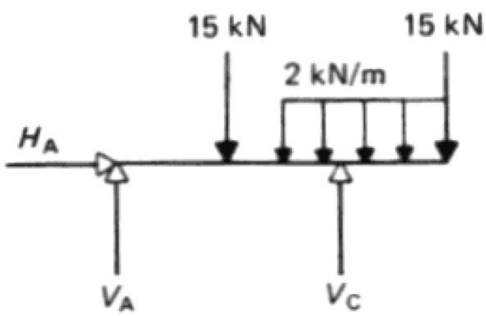
Step 15: Click on **Results>Reactions>Reaction Forces/Moments** and select the load combination COMB 1. Select FXYZ. Check on Values and click on the box … next to Values to change number of decimal points to 2 and click OK to see reactions graphically.



Step 16: Click on **Results>Result Tables>Reactions** and select the load combination COMB 1. Click OK to display the reactions in the table format. Note the Summation of all forces from the Reaction Table.



Hand Calculations:



(1) To determine H_A

$(\Sigma H = 0)$ There are no horizontal loads. $\therefore H_A = 0$

(2) To determine V_C

Take moments about A:

Note that the moment of the UDL (Uniformly Distributed Load) is the resultant total of UDL (2 x 3 = 6 kN) multiplied by the distance from A to the line of action of that resultant (i.e. 4.5 m)

$(\Sigma M_A = 0)$
$+(15 \times 2) - (V_C \times 4) + (2 \times 3 \times 4.5) + (15 \times 6) = 0$
$\therefore V_C = +36.75\ kN$

(3) To determine V_A
$\Sigma V = 0$

$$+V_A - 15 + V_C - (2 \times 3) - 15 = 0$$
$$+V_A - 15 + (+36.75) - 6 - 15 = 0$$
$$\therefore V_A = -0.75\ kN\ (i.e.\ 0.75\ kN\ downwards)$$

(4) Check by taking moments about C:
$$\Sigma M_C = +(V_A \times 4) - (15 \times 2) + (2 \times 3 \times 0.5) + (15 \times 2)$$
$$= +(-0.75 \times 4) - 30 + 3 + 30 = 0$$
$$\therefore Correct$$

Comparison of Results

Unit : kN

Reaction	Node Number	Theoretical	Midas Gen
H_A	1	0.00	0.00
V_A	1	-0.75	-0.75
V_C	3	36.75	36.75

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 1.2, Page 6.