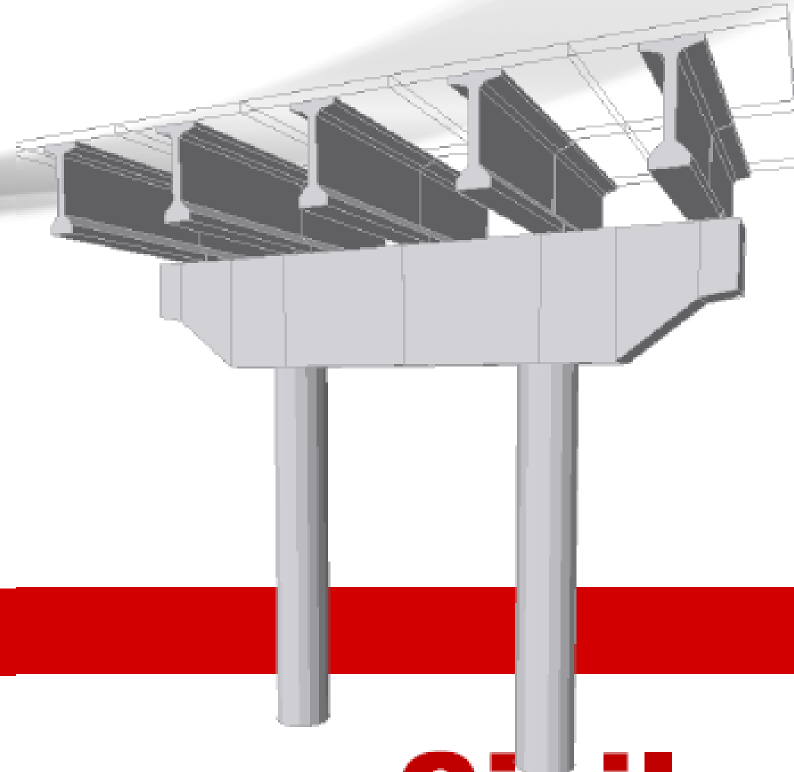
Pier Modeling Example



Bridging Your Innovations to Realities

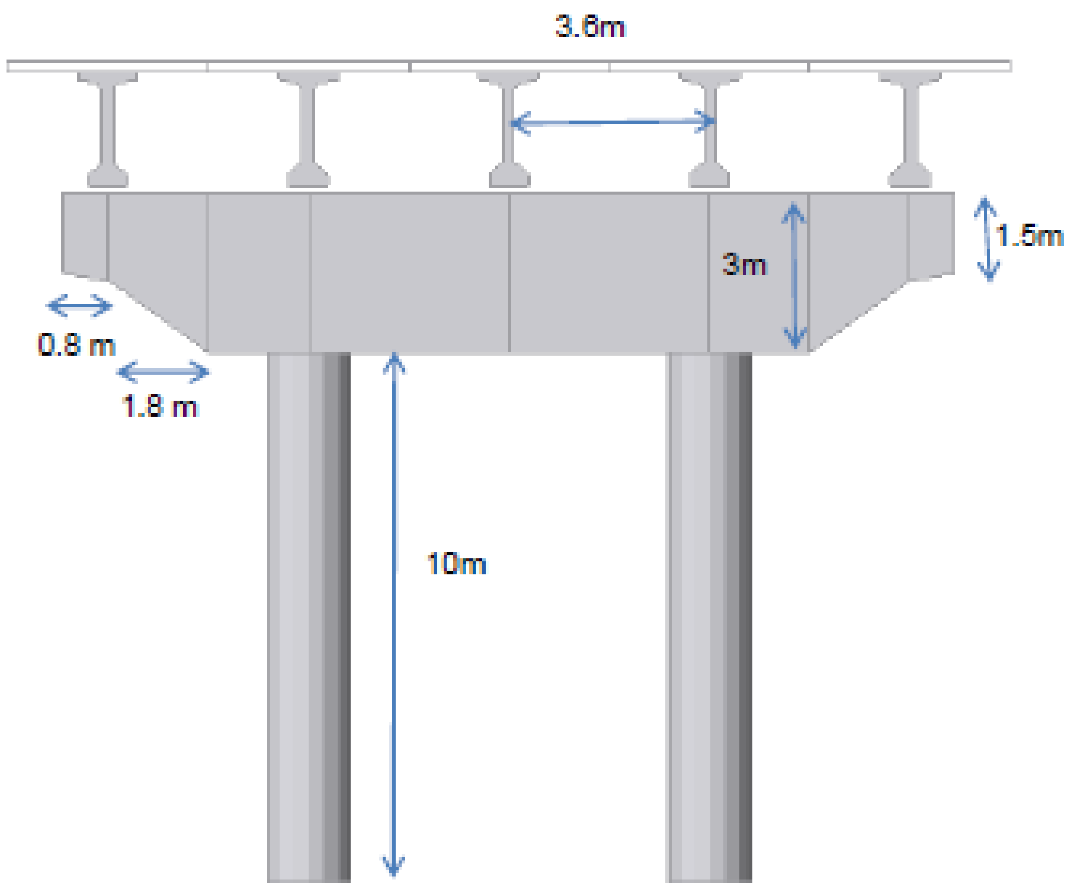
midas Civil

Training material

Overview: Pier Modeling

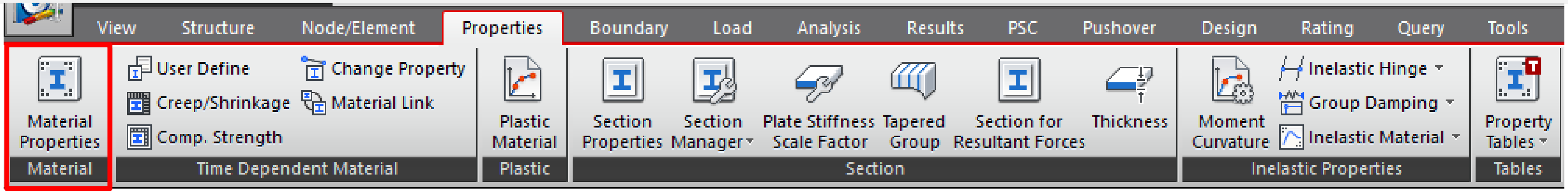
1. **Response Spectrum Analysis**
    1. Introduction
    2. Material
    3. Sections
    4. Modeling

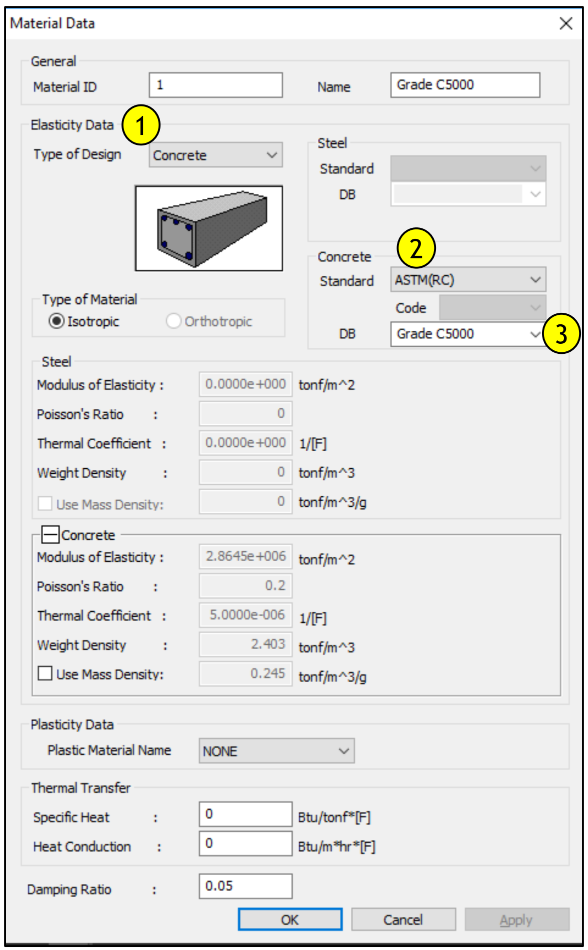
1. Introduction: Pier Modeling



Front View

2. Material



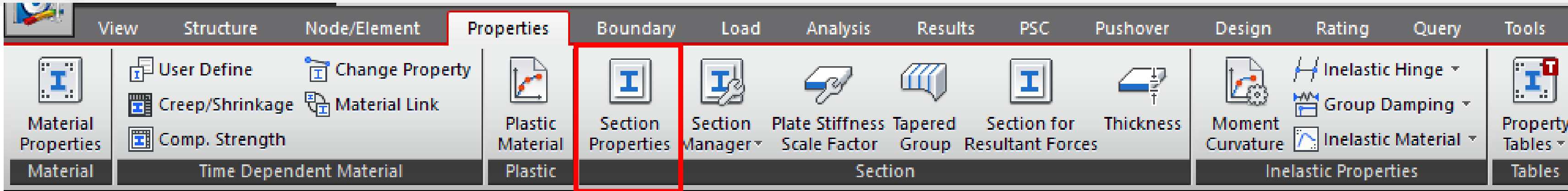


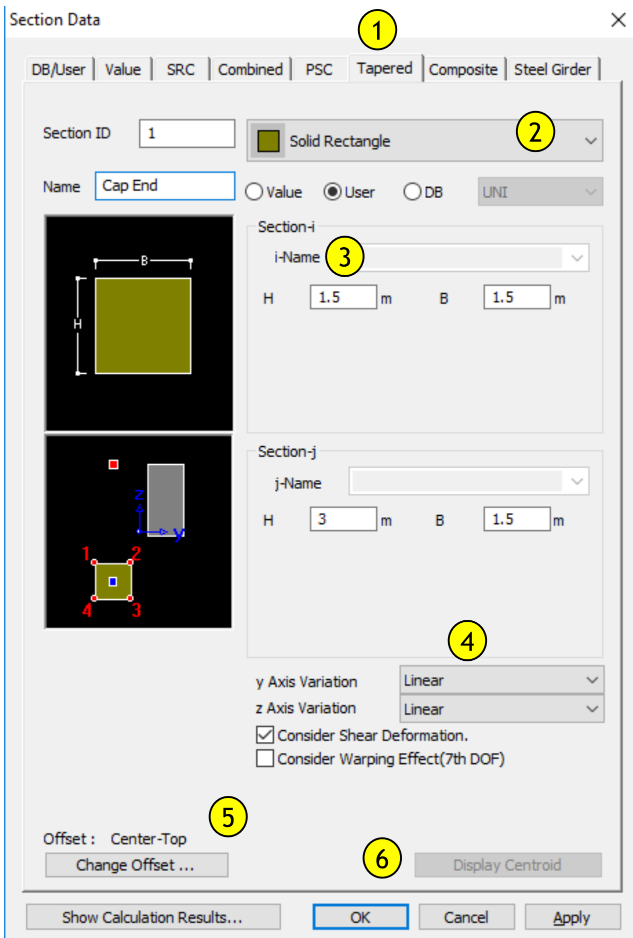
Properties > Material Properties

Concrete:

1. *Select Concrete under type of design.*
2. *Select ASTM(RC) under Concrete Standard.*
3. *Select C5000*

3. Sections





Properties > Section Properties

Add Section

Cap End:

1. Select Tapered Section Tab.

2. Select Solid Rectangle section for the ends of Pier Cap.

3. Enter values for i and j .

4. Apply y and z axis variation for linear.

5. Set Offset at Center Top

6. Click OK

Cap Mid:

1. Select DB/USER tab

2. Select Solid Rectangle

3. Enter 3 m X 1.5 m

4. Set Offset at Center Top

5. Click Apply

Pier:

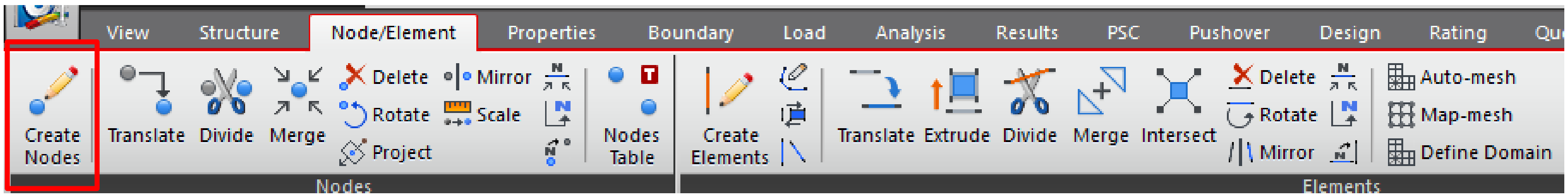
1. Select Solid Round

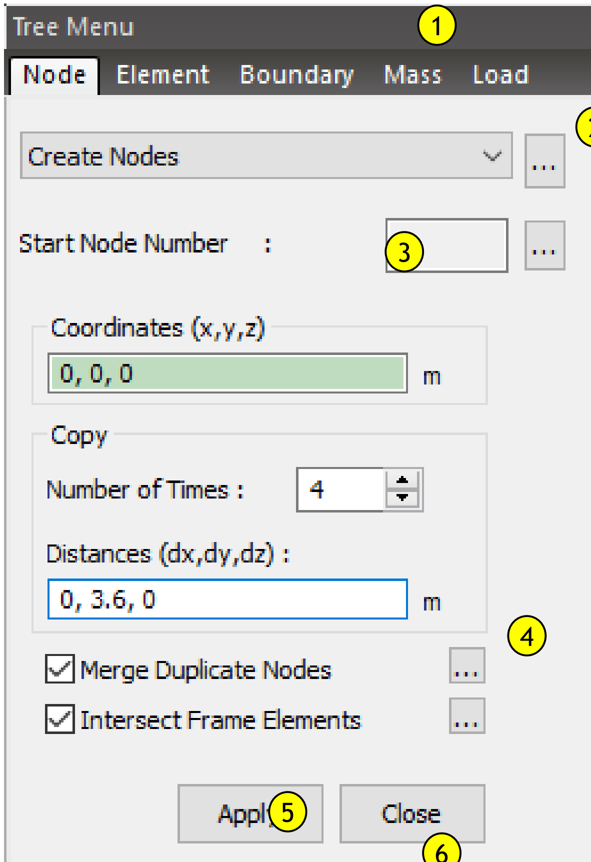
2. Enter diameter as 1.5 m

3. Set Offset at Center - Center

4. Click OK

4. Modeling



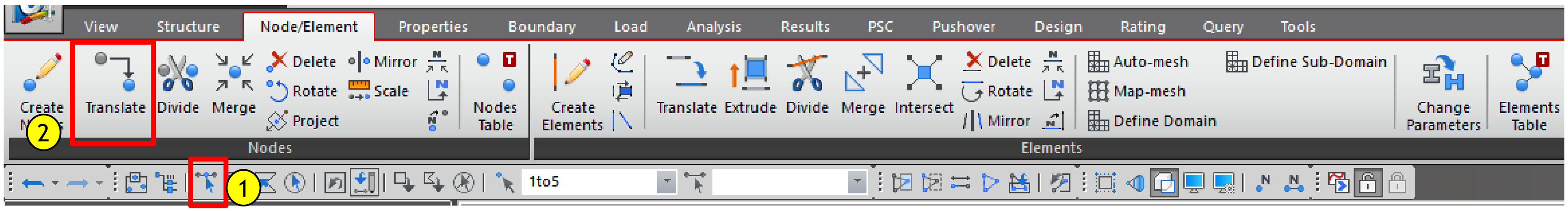


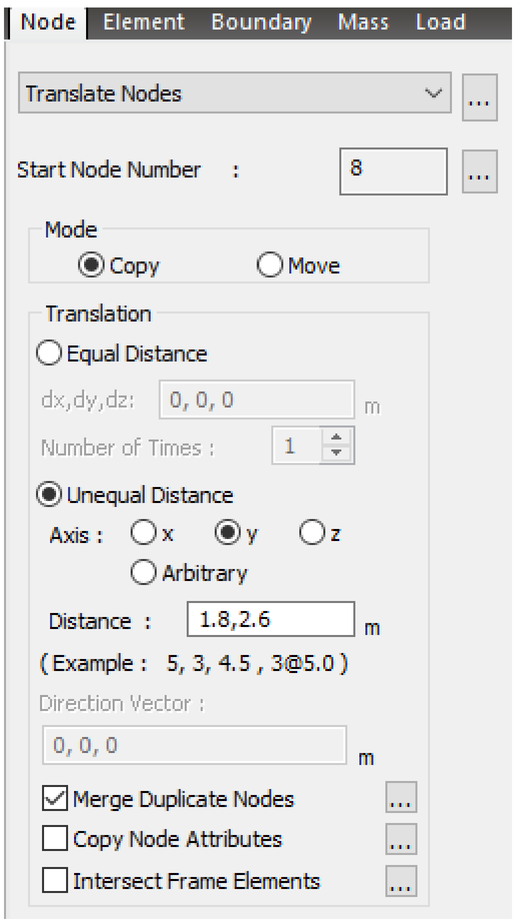
Node/Element > Create Nodes

Add Section

1. *Create Nodes and Copy node at origin 4 times by 0,3.6,0 (m)*
2. *[Zoom fit]*

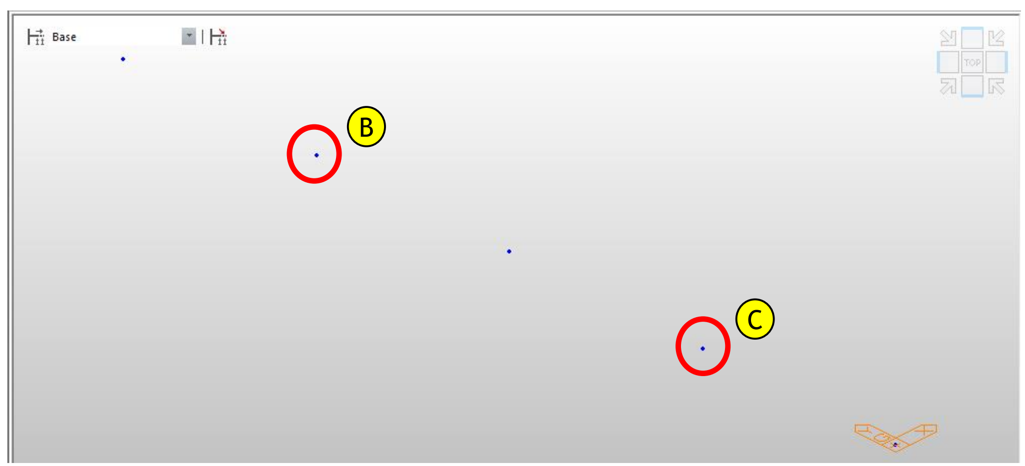
4. Modeling



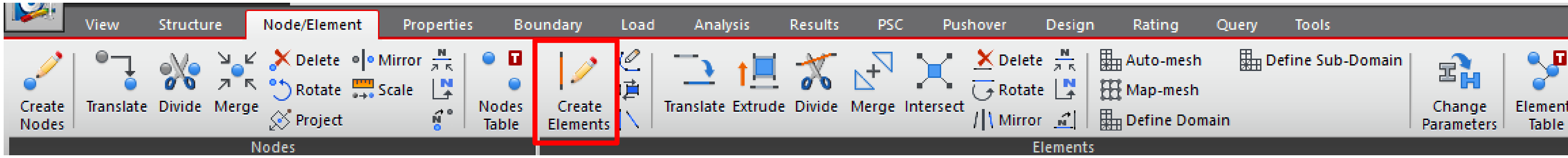


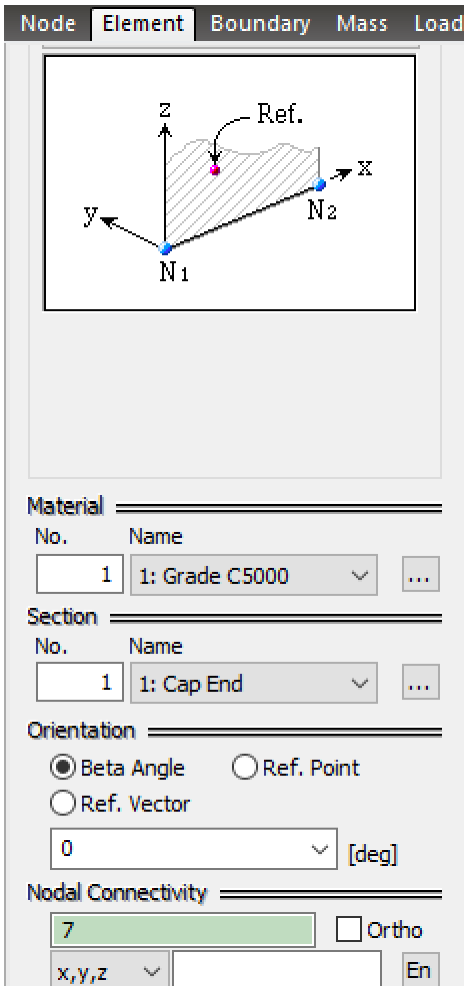
Node/Element > Translate Nodes

1. *Select Node B [using Select single]*
2. *Translate Node in circle B using the Copy function.*
3. *Repeat with Node C but distance as -1.8,-2.6 m*



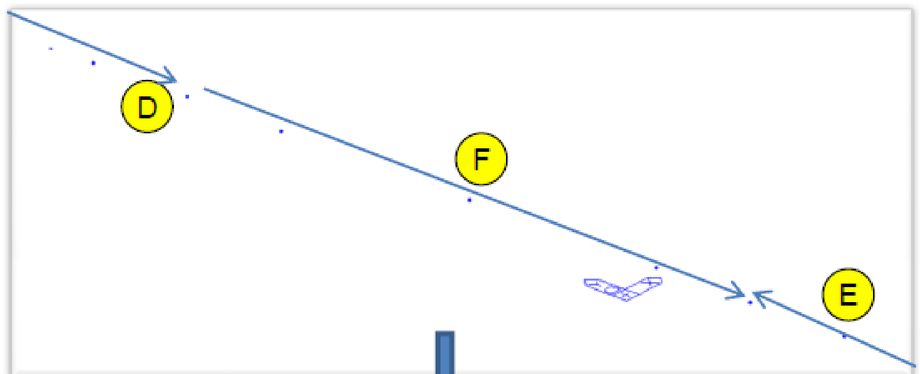
4. Modeling

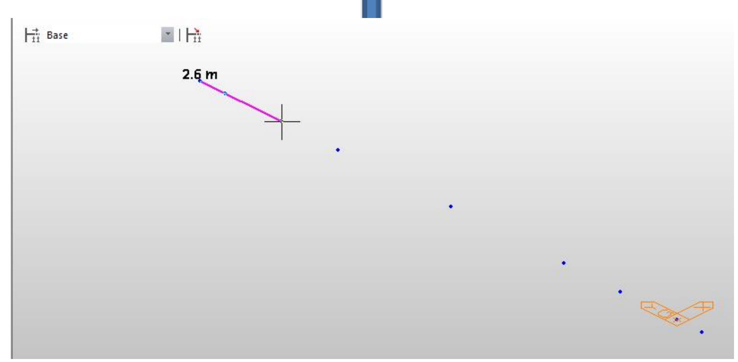




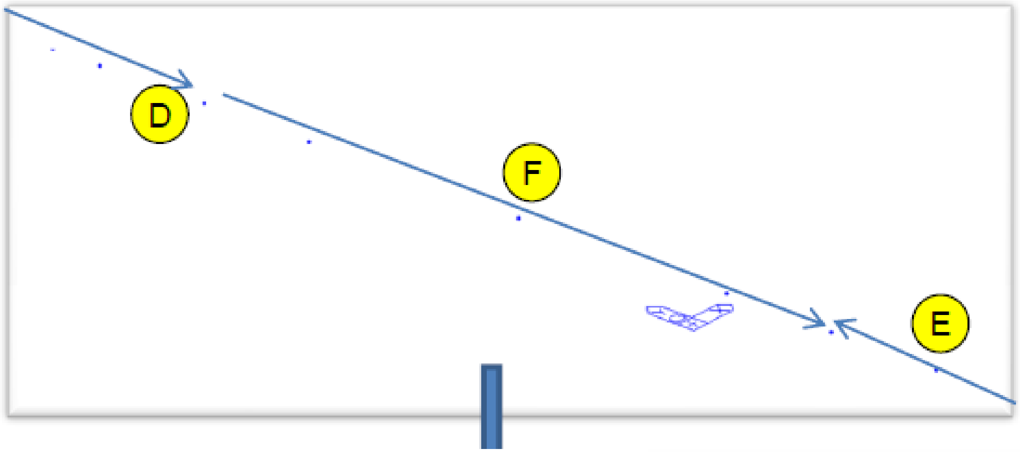
Node/Element > Create Elements

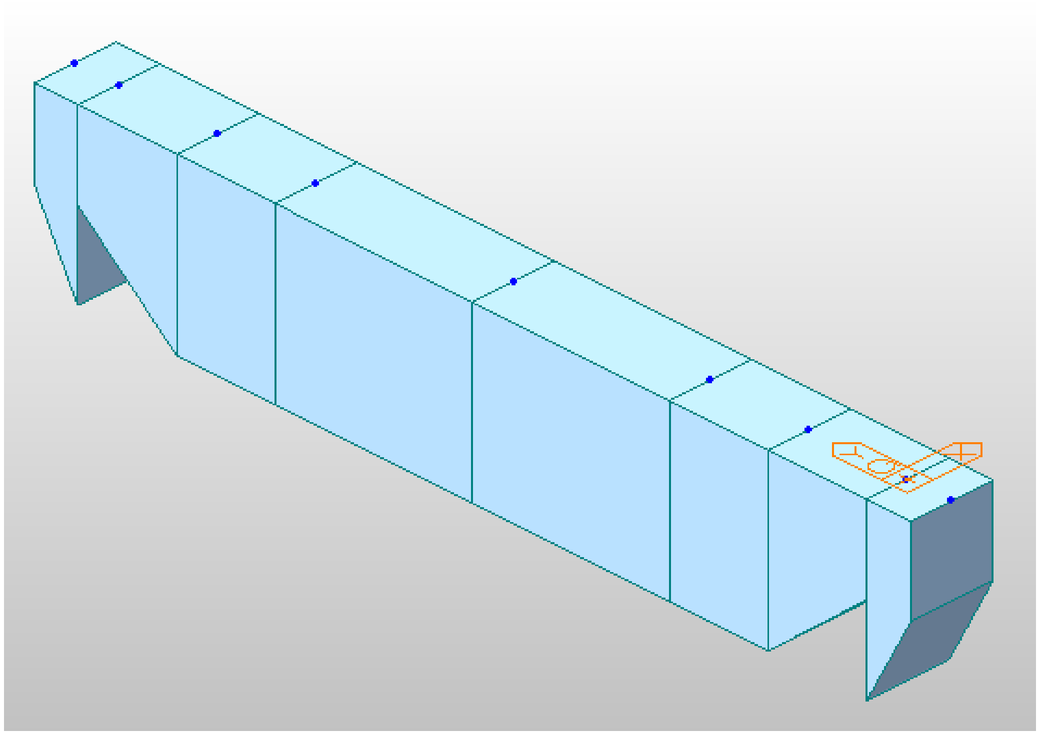
1. *Click in the nodal connectivity field to make it active and the click on the extreme nodes on either side in the direction of arrow. (to make pier cap ends) [First D then E]*
2. *Change section in 1 to Cap Mid and Click at the middle five nodes [F]*



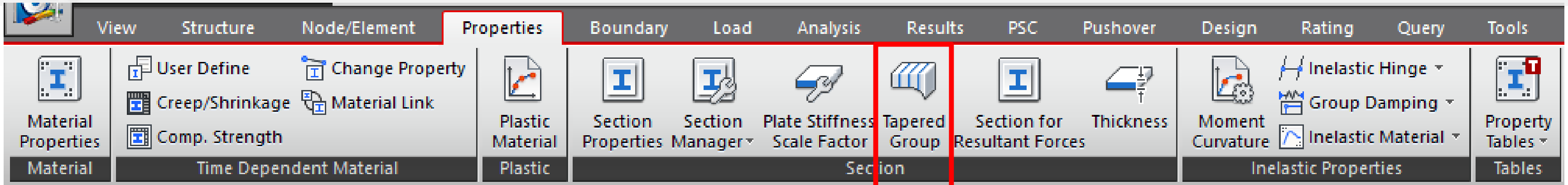


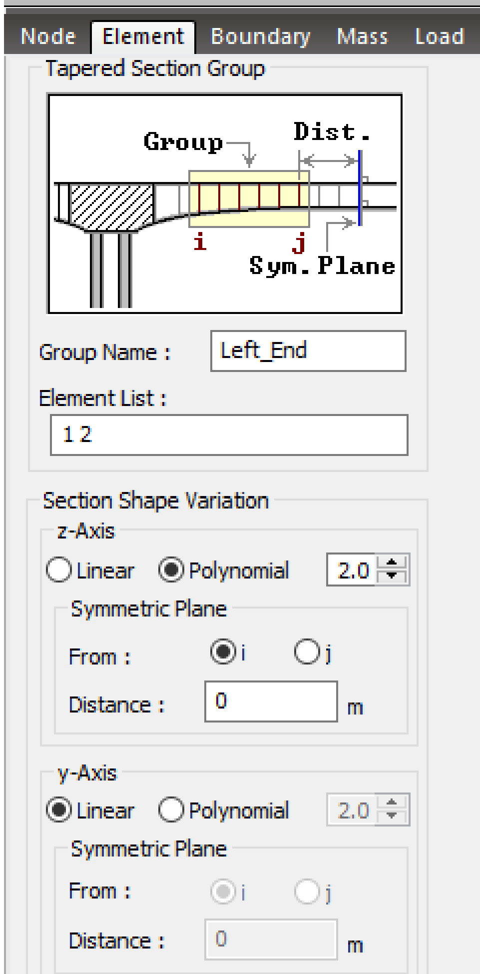
4. Modeling





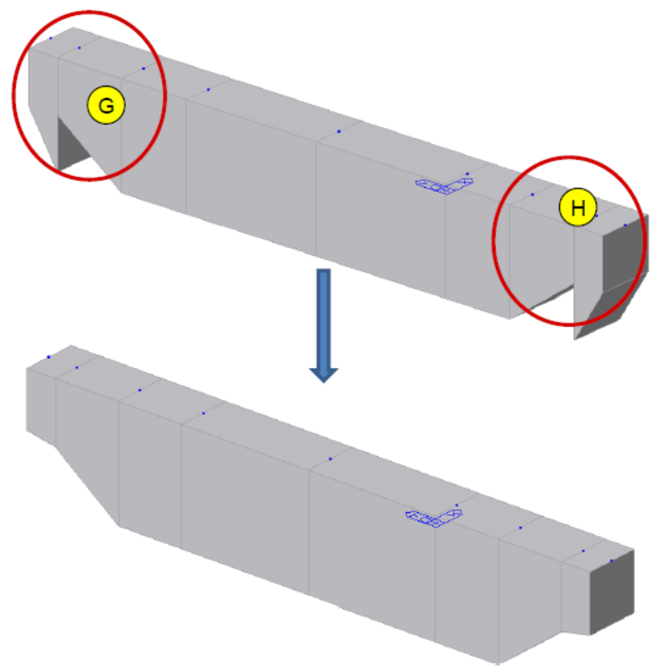
4. Modeling



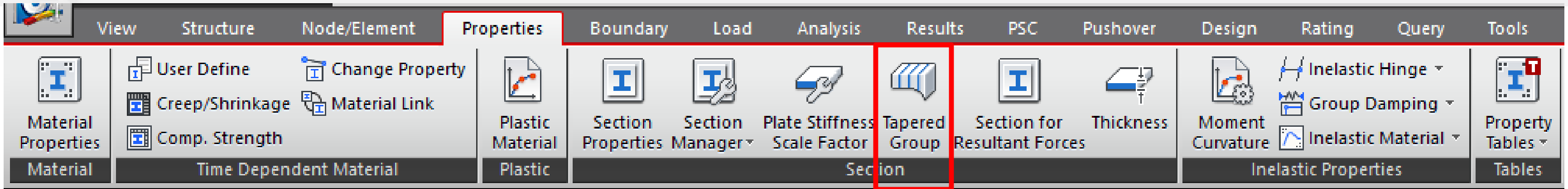


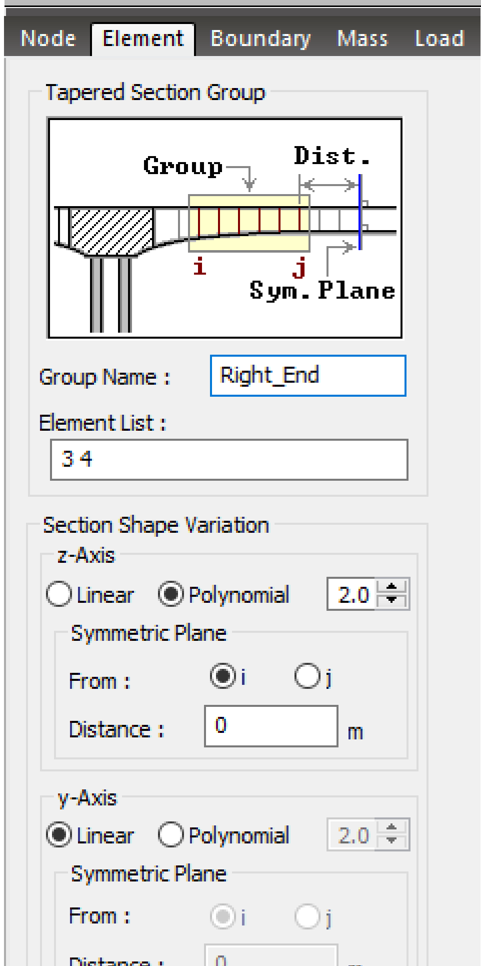
Properties > Tapered Group

1. *Select elements in group G [using single select] and enter values based on dialog box 1.*
2. *Repeat 2 with elements in group H and symmetric plane for Z axis from j (in dialog box 1)*



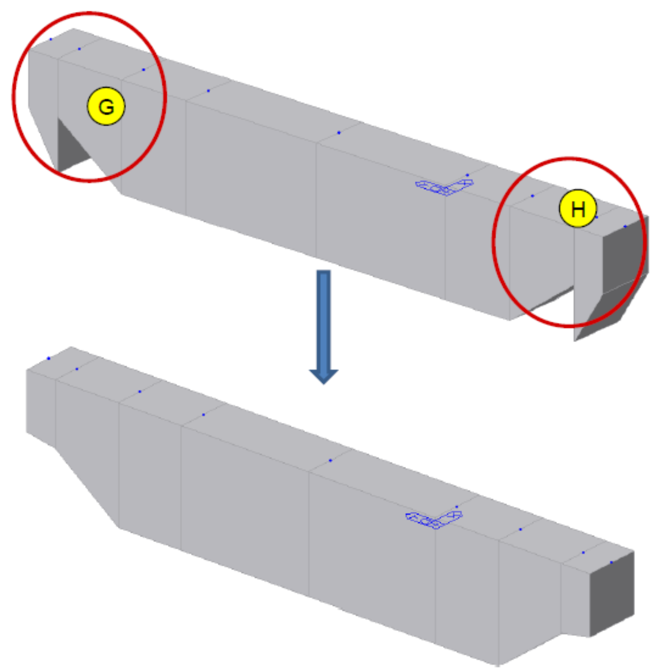
4. Modeling



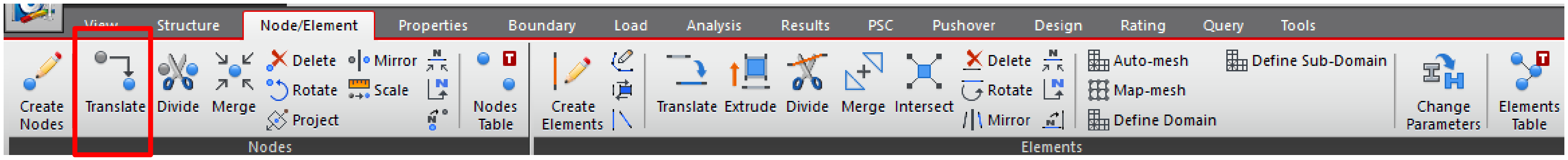


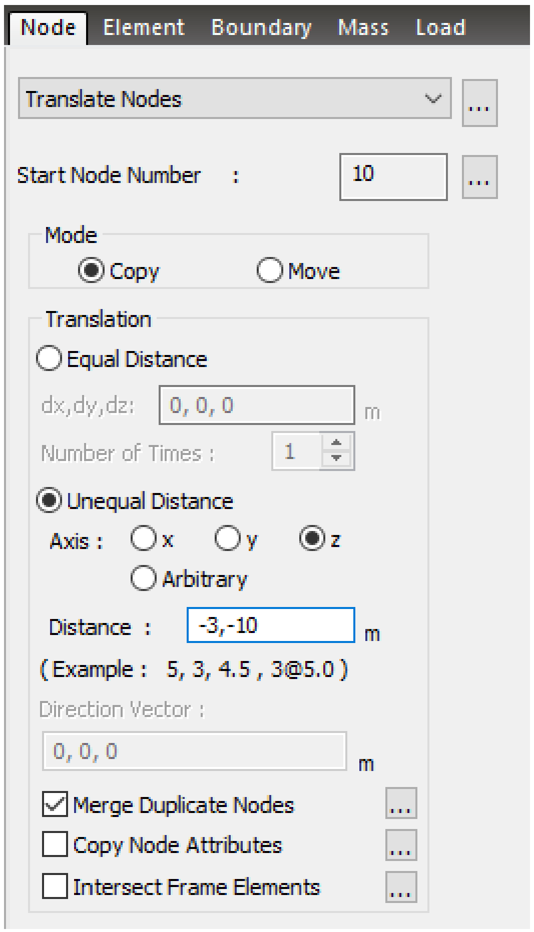
Properties > Tapered Group

1. *Select elements in group G [using single select] and enter values based on dialog box 1.*
2. *Repeat 2 with elements in group H and symmetric plane for Z axis from j (in dialog box 1)*



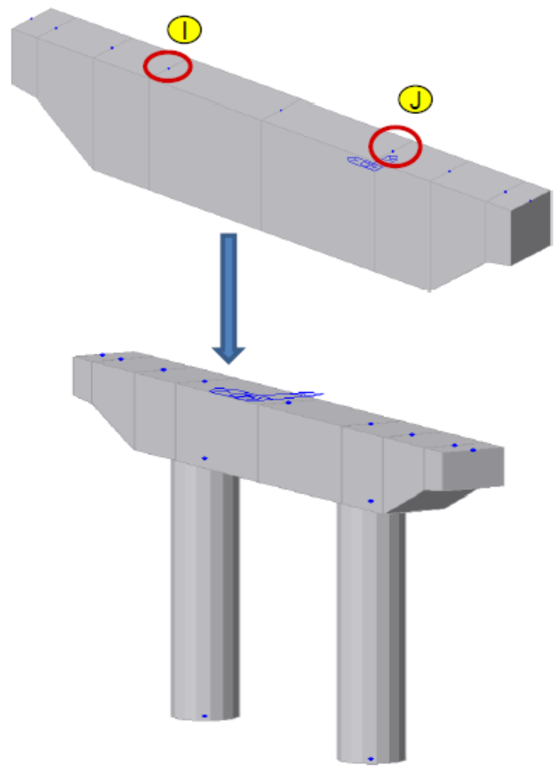
4. Modeling



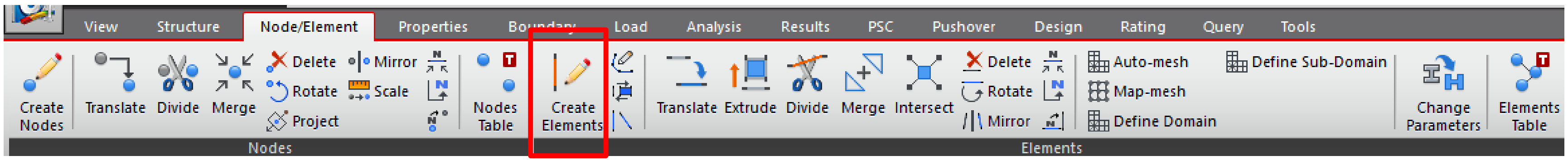


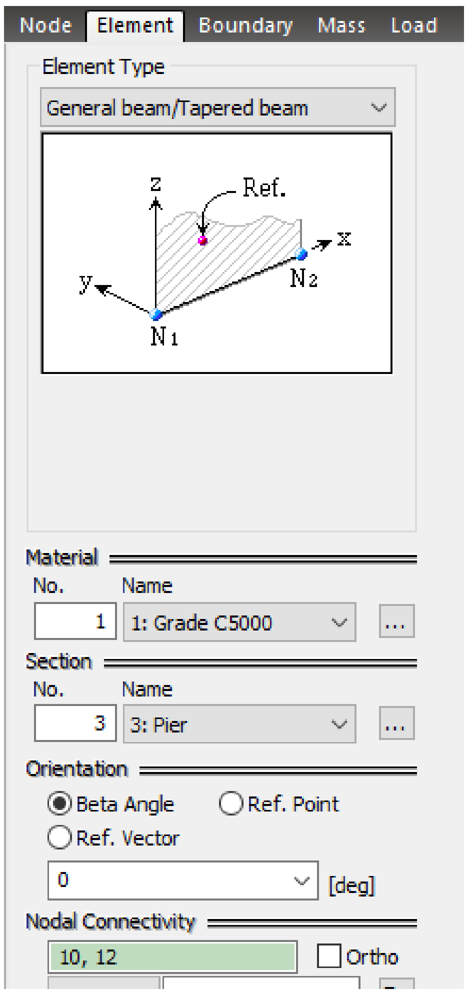
Node/Element>Translate

1. *Translate Nodes. Select Node I and J using .*
2. *2. Model>Elements> Create Elements. Create pier by clicking at the newly create nodes (after clicking in Nodal connectivity field)*



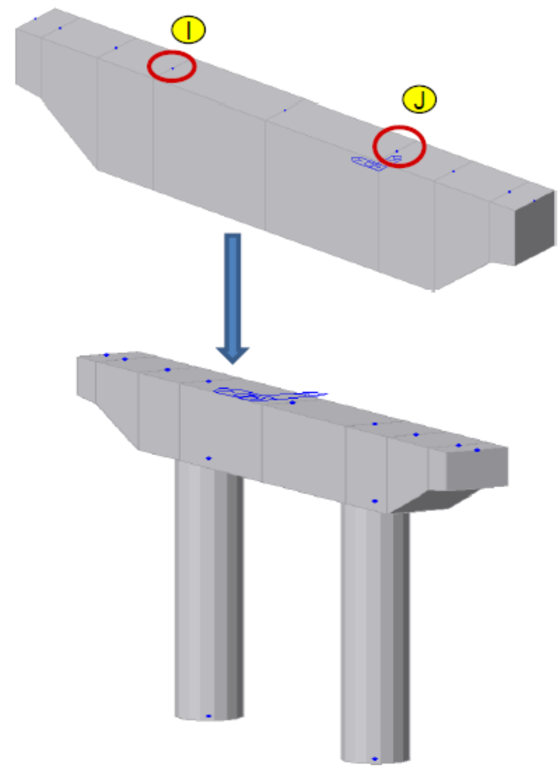
4. Modeling



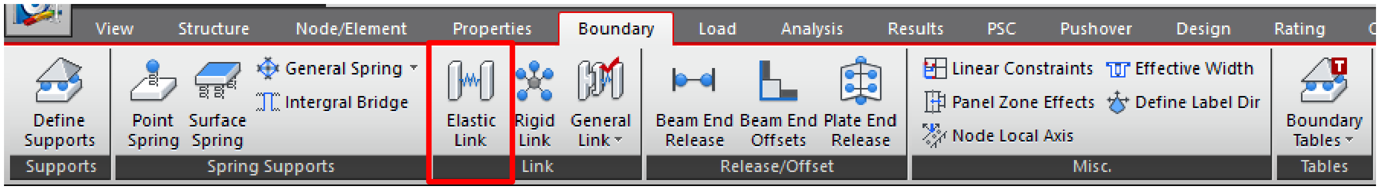


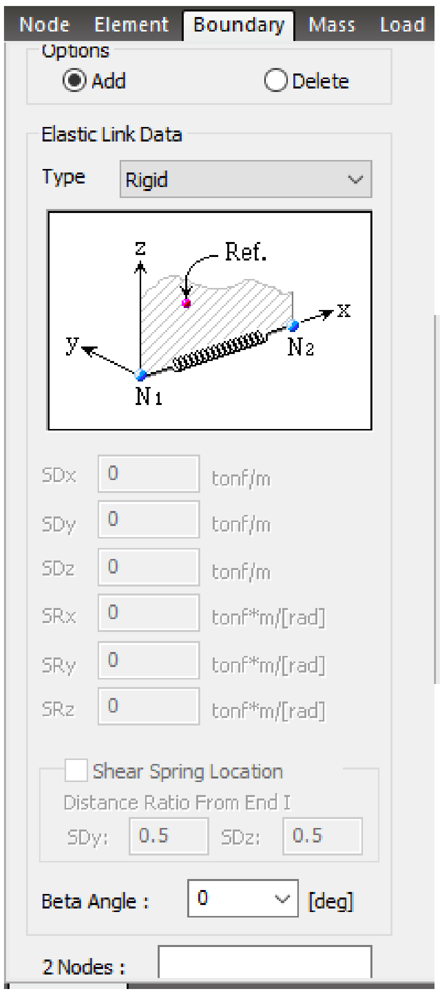
Node/Element>Translate

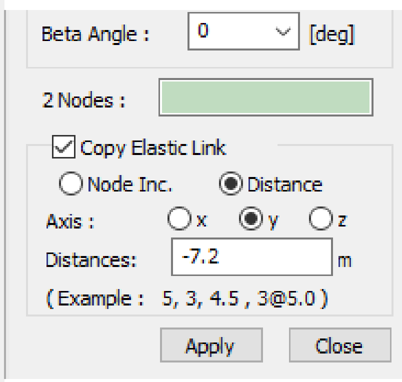
1. *Translate Nodes. Select Node I and J using .*
2. *2. Model>Elements> Create Elements. Create pier by clicking at the newly create nodes (after clicking in Nodal connectivity field)*



4. Modeling

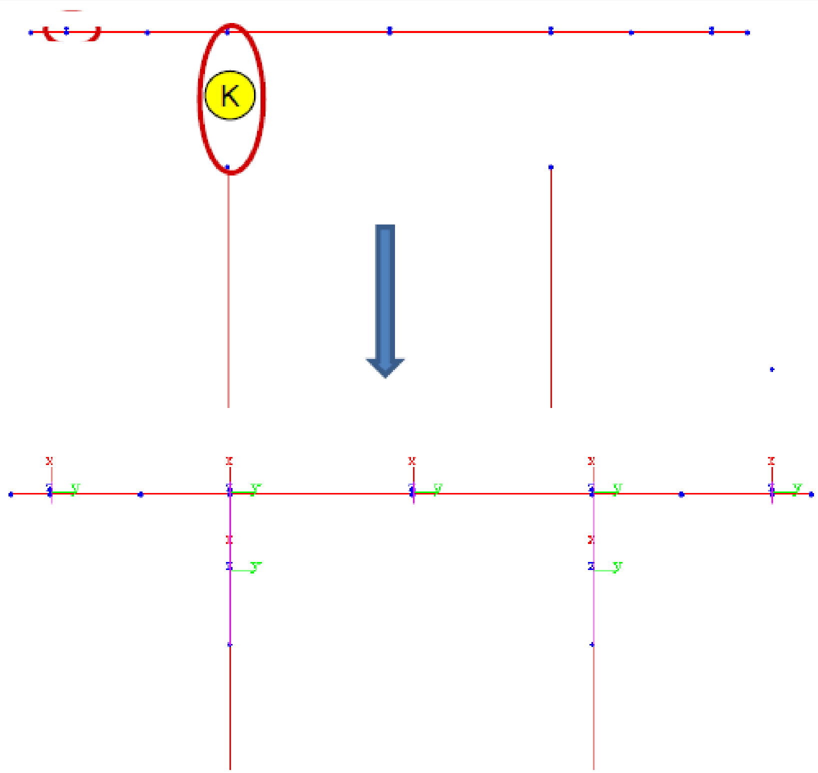




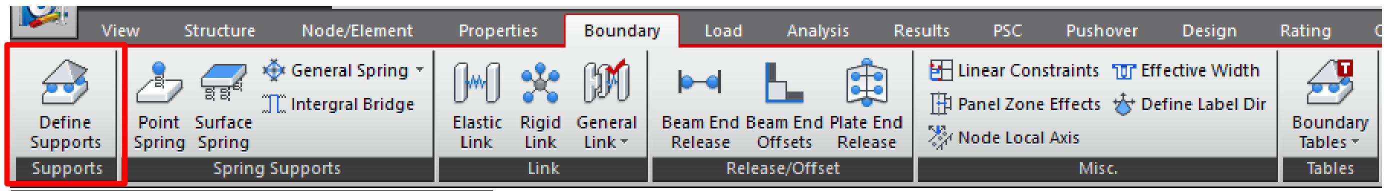


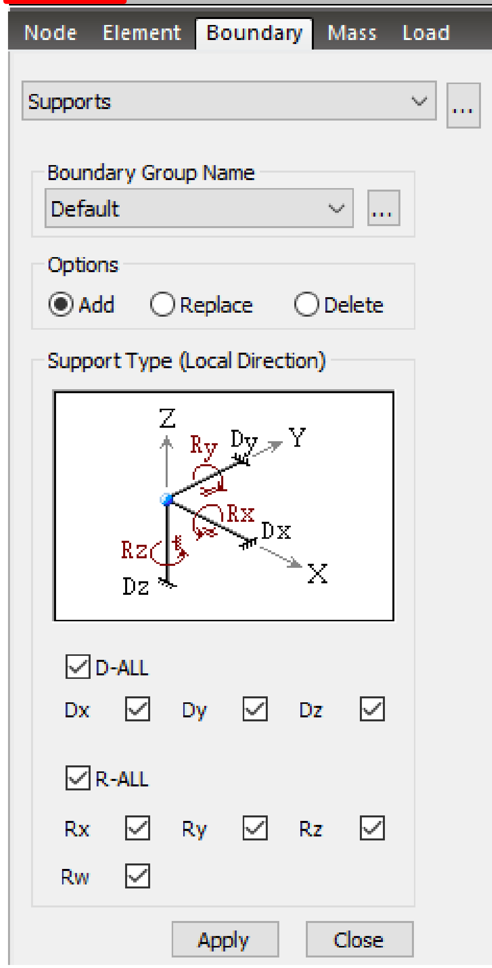
Boundary > Elastic Link

1. *Activate All , Hidden off (to see wire frame). Select Left View.*
2. *Pier cap connection: Check on Copy and enter the distance. Then click in the field against 2 Nodes and click on nodes in group K*



4. Modeling

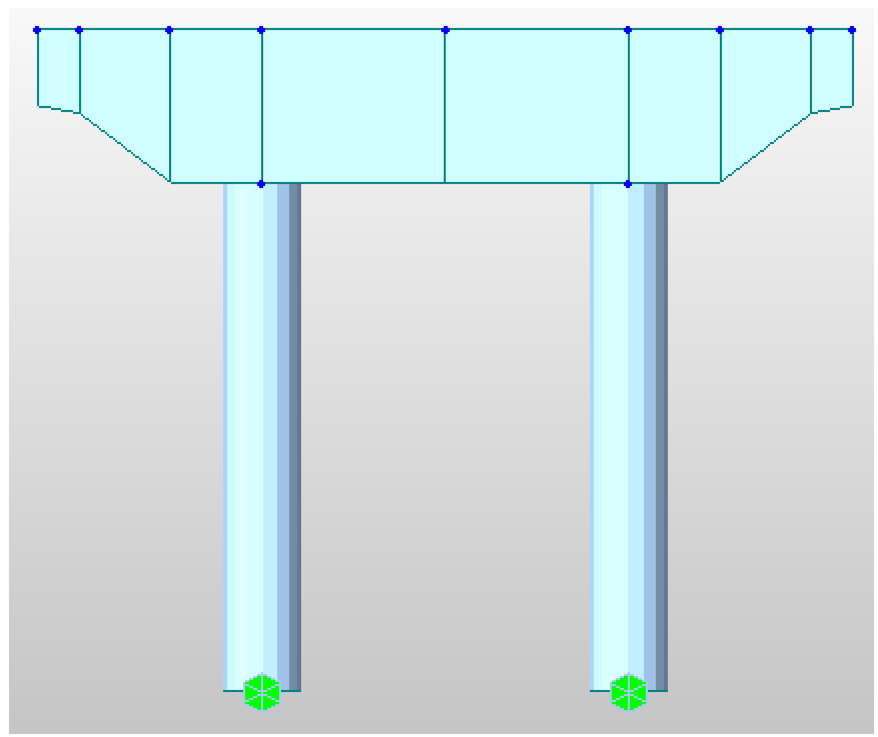




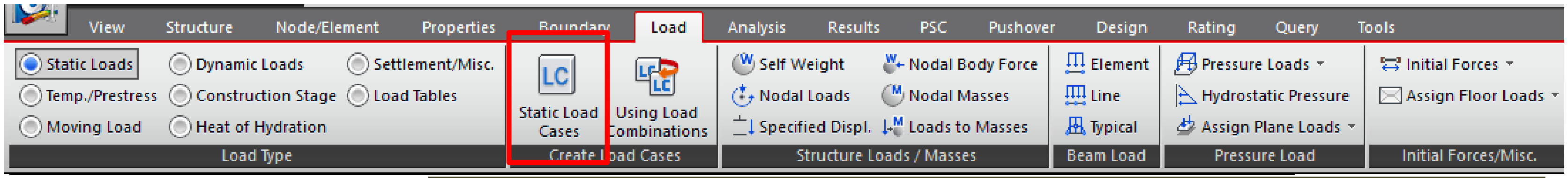
Boundary > Define Supports

1. Select the nodes at the bottom of the piers to define the support conditions

2. D-All and R-All to restrain all the degrees of freedom and define a fixed condition.

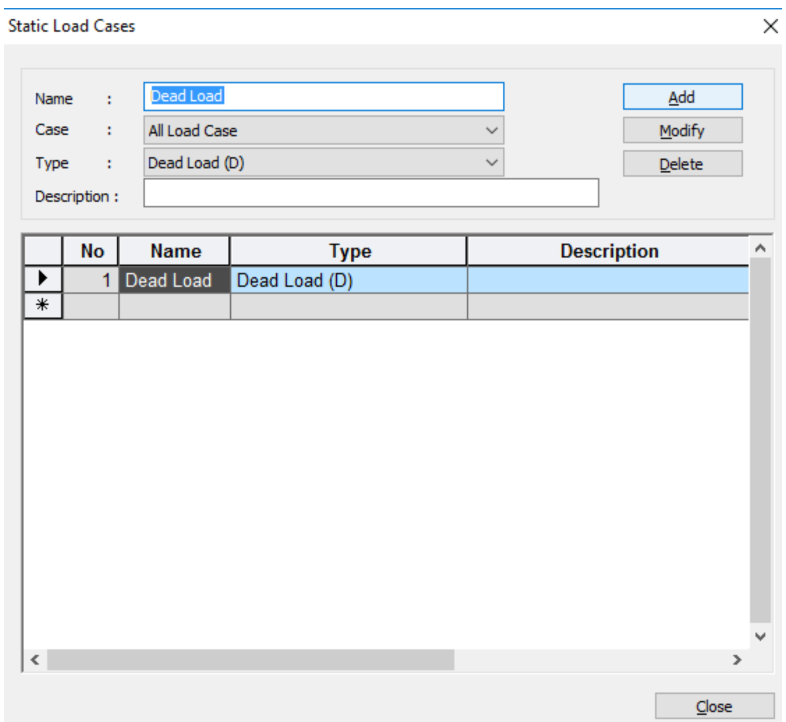


4. Modeling

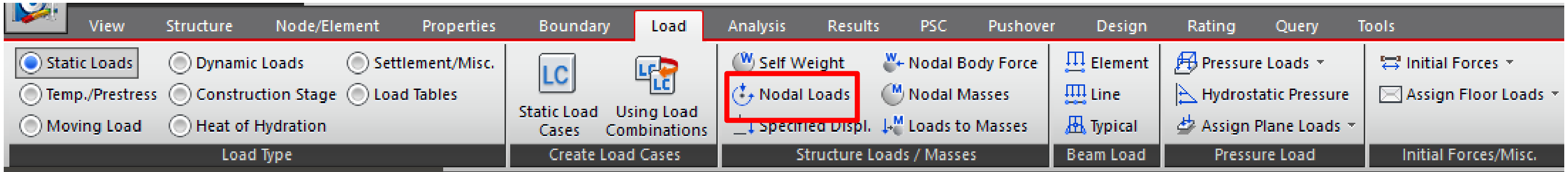


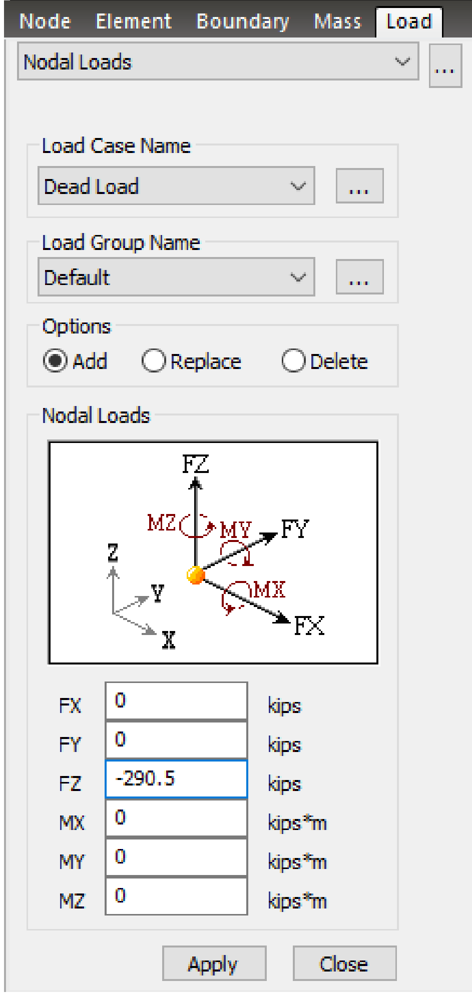
Load > Static Load > Static Load Cases

1. *Define the Dead Load case*
2. *Type Dead Load (D)*
3. *Click Add*



4. Modeling

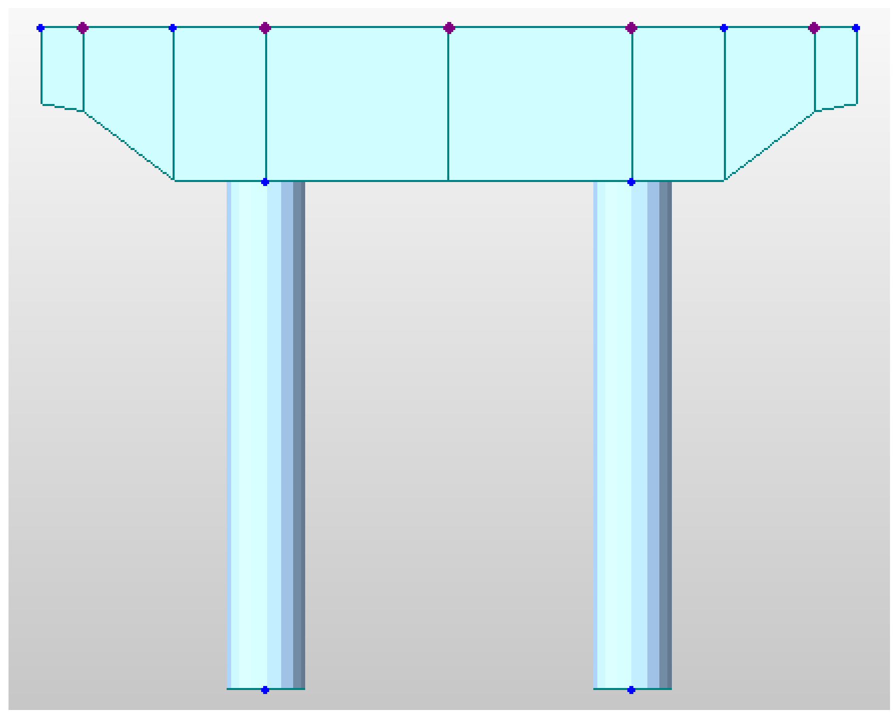




Load > Static Loads > Nodal Loads

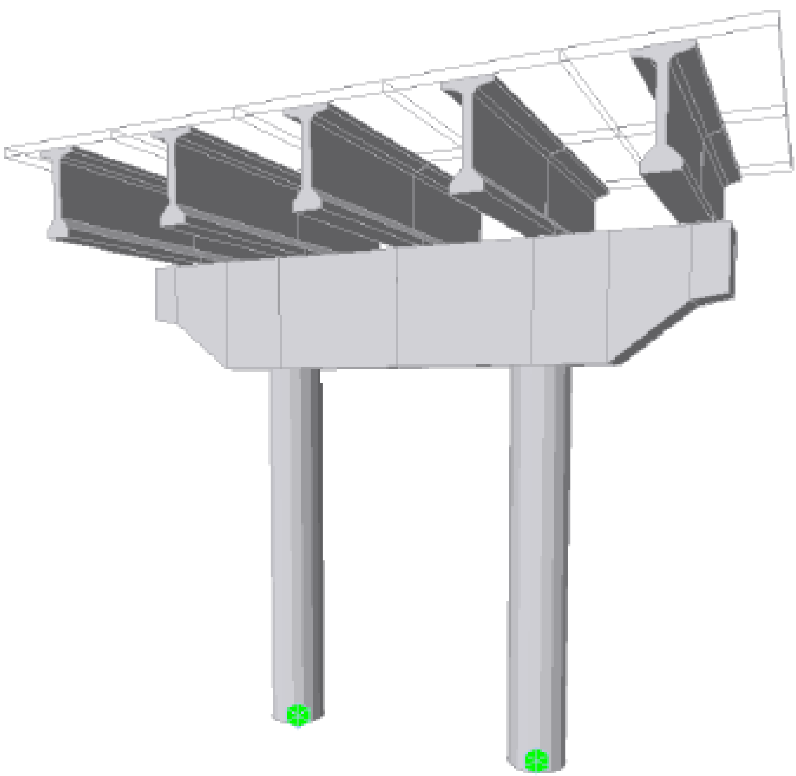
1. Select the nodes shown to apply the dead load as nodal loads

2. Apply -290.5 kips in the FZ (Vertical direction)



4. Modeling

After pier modeling, support conditions based on requirements can be applied. In this example, a small section of the bridge was taken. The bridge span can be extended and other piers can be modeled based on the approach outlined in this tutorial.



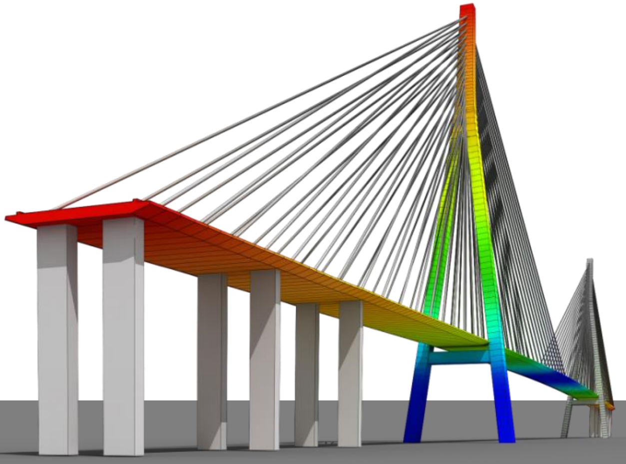
Pier Modeling Example

Bridging Your Innovations to Realities

Training material

MIDAS Technical Support
http://globalsupport.midasuser.com/

Section Design as per AASHTO-LRFD 2012 using midas GSD



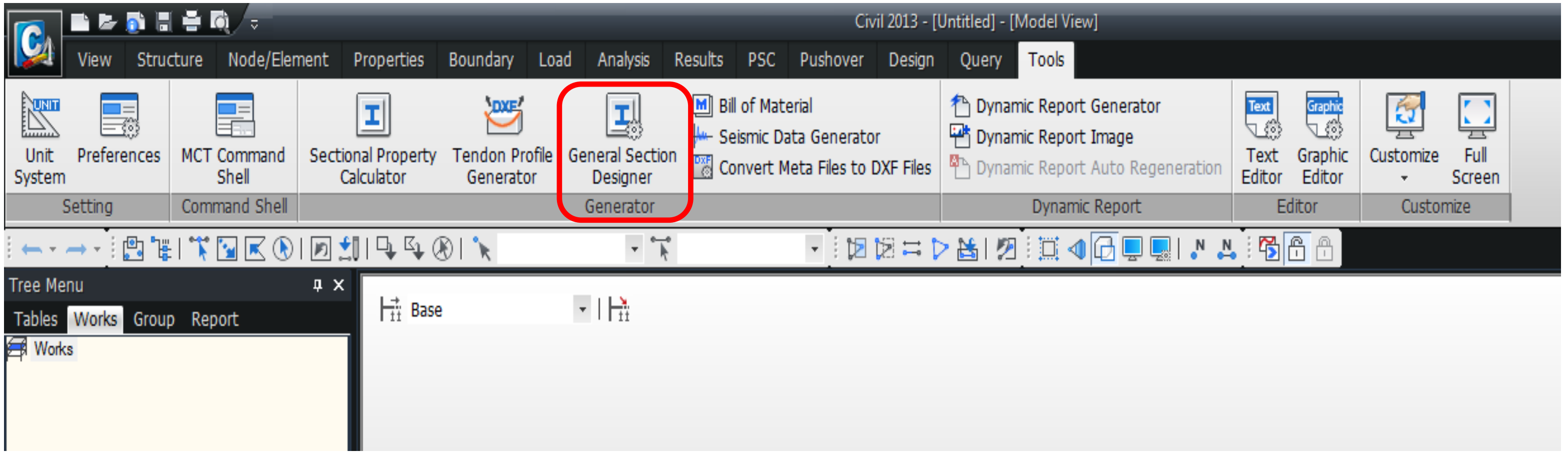
Bridging Your Innovations to Realities

Introduction

In this tutorial, it's shown how to design an irregular hollow reinforced concrete as per AASHTO-LRDF 2012. The section is designed using midas GSD (General Section Designer), which is an add-on module available with midas Civil and midas Gen. Nonlinear material model Mander model for Concrete and bilinear curve for steel are used to meet the AASHTO code requirement. The software develops 2D & 3D interaction diagram and moment curvature curve (with idealized curve), and stress contour for both concrete and steel rebar. This module can be linked to midas Civil and Gen and information (like section properties, materials properties and load combinations) can be exchanged.

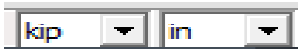
Start

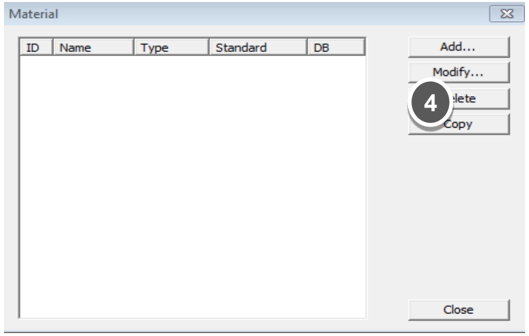
1. Open Midas Civil
2. Under Tool tab click General Section Designer to open a new midas GSD

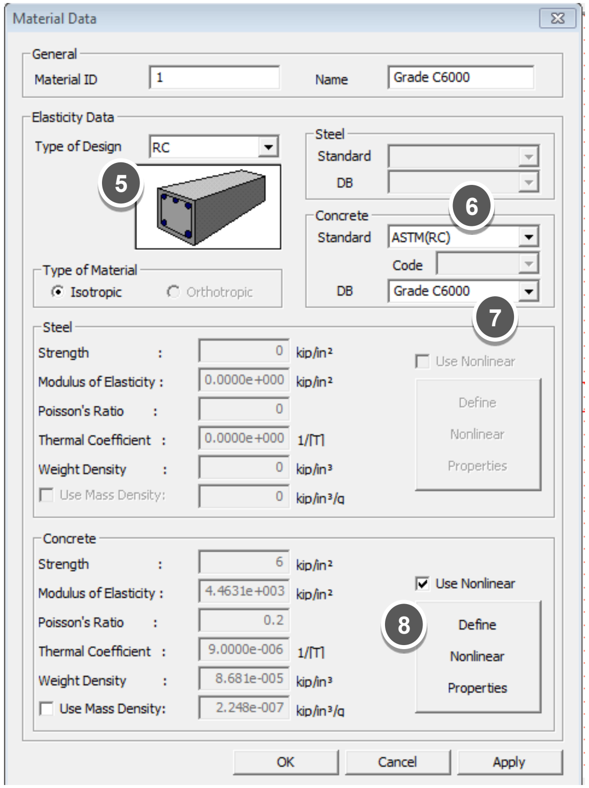


Material

1. In the midas GSD:
2. At the right bottom corner, change units to *kips* and *in*
3. Model > Material
4. Click > Add
5. Select RC from Type of Design
6. Select > ASTM (RC) from Standard
7. Select Grade 6000 from DB
8. Check on Use Nonlinear and click Define Nonlinear Prop.

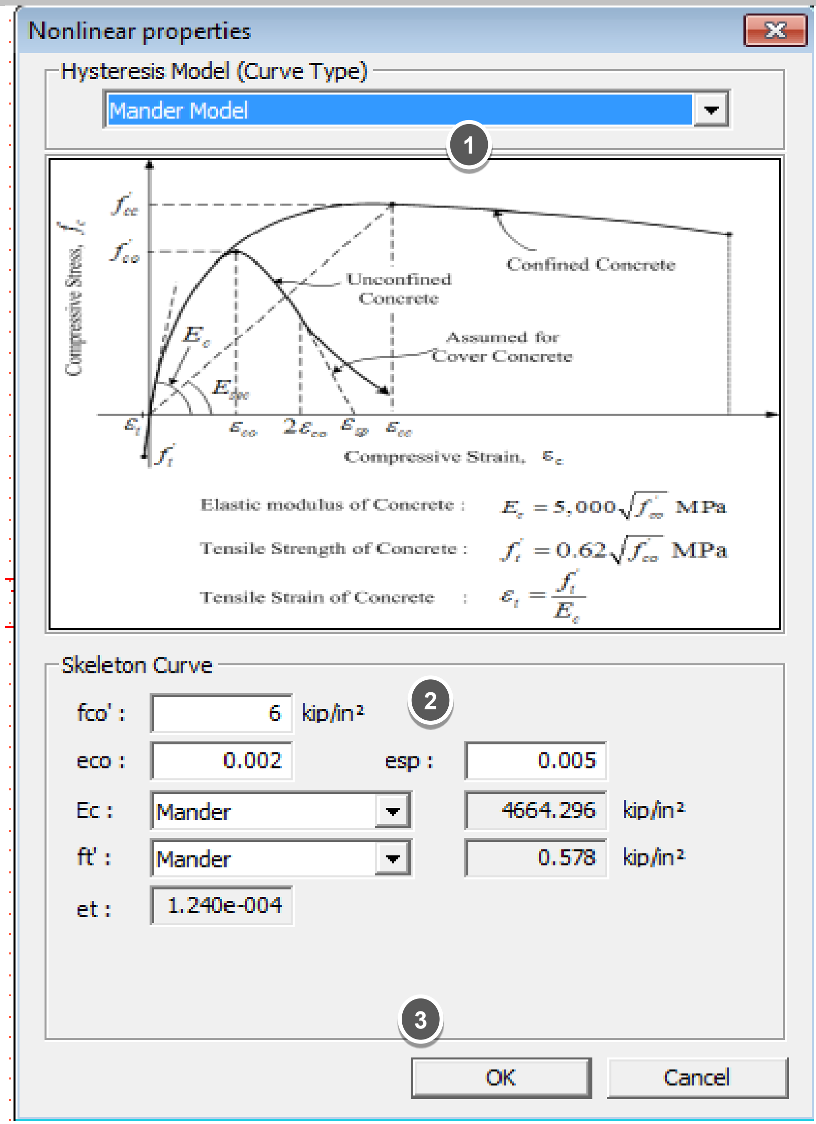






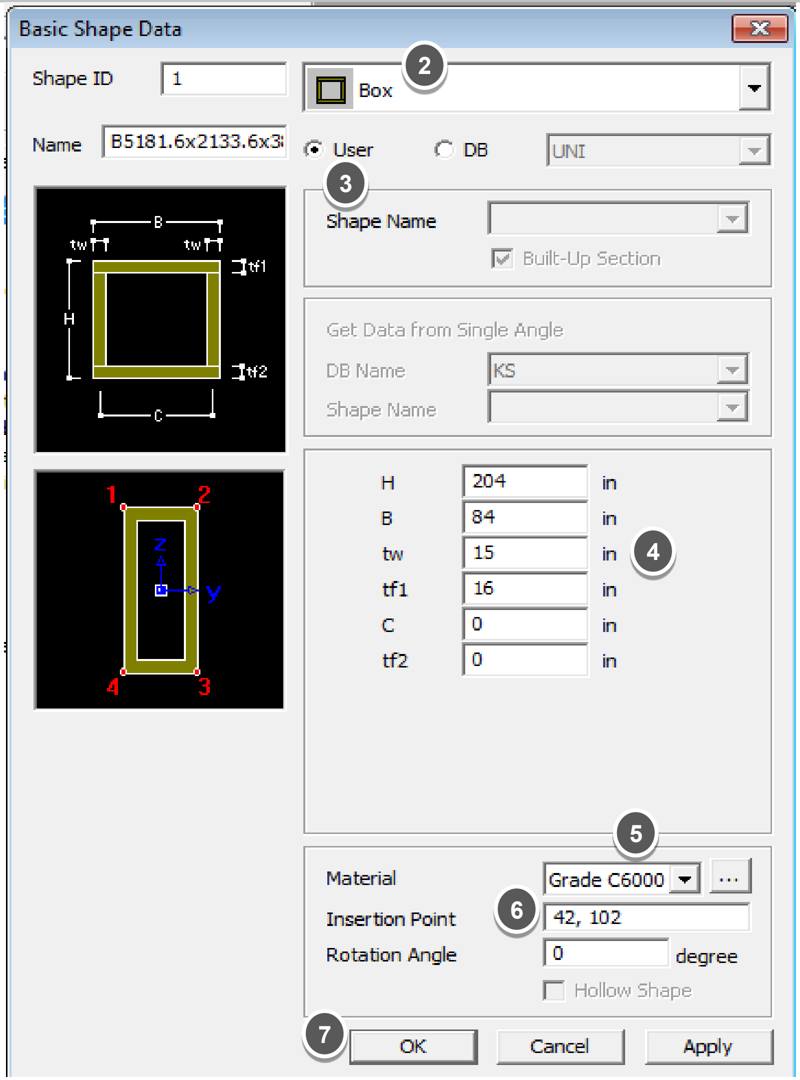
Material

1. Select Mander Model from
   Hysteresis Model menu
2. Ensure the values in Skeleton
   Curve section are as shown in the picture
alongside, *these are selected based on defined*
   material, but you can change if required
3. Click > OK
4. Click >OK
5. Close



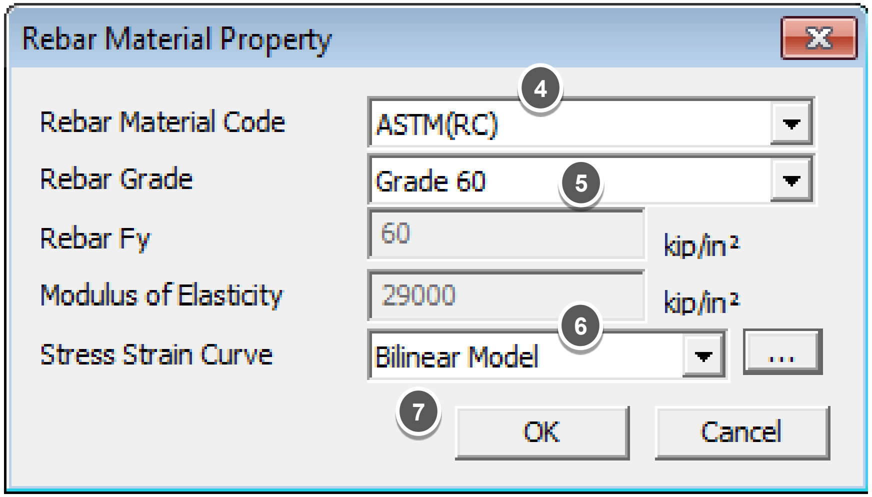
Geometry

1. Model > Shape > Basic Shape
2. Select Box from dropdown menu
3. Select >User
4. H: 204, B: 84, t_w: 15, t_{f1}: 16 in
5. Material: Grade C6000
6. Insertion Point: 42, 102
7. Ok
8. Click  to see entire new shape



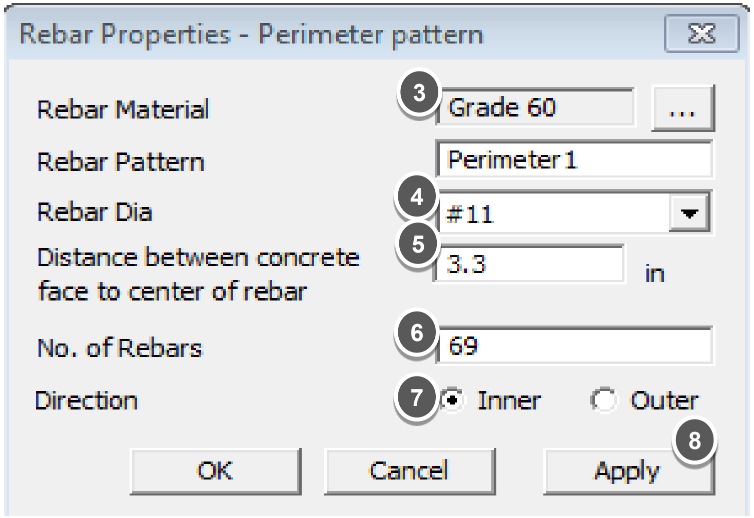
Reinforcement

1. Model > Rebar > Rebar Material Property …
2. Select ASTM
3. Grade 60
4. Select Bilinear Model
5. Ok



Reinforcement

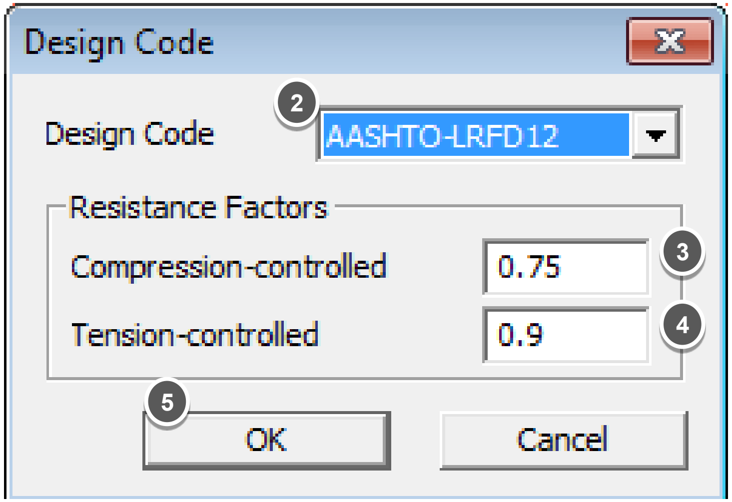
1. Model > Rebar > Rebar- Perimeter Pattern
2. Select Shape 1 from tree menu
3. Rebar Material : Grade 60
4. Rebar Dia: #11
5. Distance 3.3 in
6. No. of rebars: 69
7. Direction: Inner
8. Apply

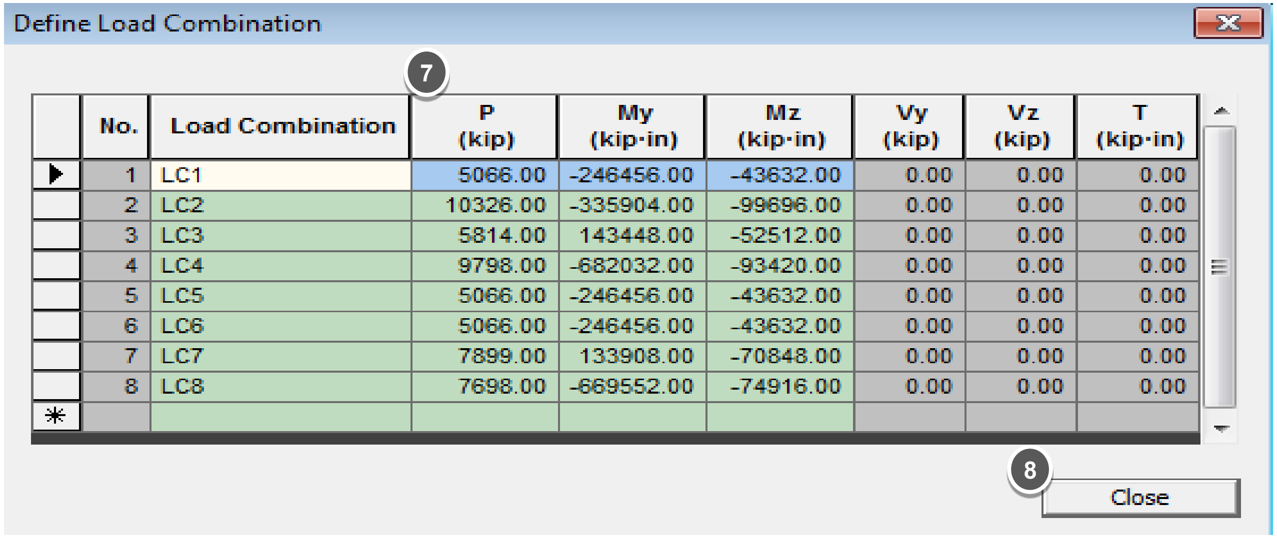


Design Code & Load Combinations

1. Option > Design Code
2. Design Code: AASHTO-LRFD 12
3. Compression-controlled: 0.75
4. Tension-controlled: 0.9
5. OK
6. Load > Load Combination
7. Enter 8 load combinations as it's shown below
8. Close
9. Click

To perform design





	No.	Load Combination	P (kip)	My (kip·in)	Mz (kip·in)	Vy (kip)	Vz (kip)	T (kip·in)
▶	1	LC1	5066.00	-246456.00	-43632.00	0.00	0.00	0.00
	2	LC2	10326.00	-335904.00	-99696.00	0.00	0.00	0.00
	3	LC3	5814.00	143448.00	-52512.00	0.00	0.00	0.00
	4	LC4	9798.00	-682032.00	-93420.00	0.00	0.00	0.00
	5	LC5	5066.00	-246456.00	-43632.00	0.00	0.00	0.00
	6	LC6	5066.00	-246456.00	-43632.00	0.00	0.00	0.00
	7	LC7	7899.00	133908.00	-70848.00	0.00	0.00	0.00
	8	LC8	7698.00	-669552.00	-74916.00	0.00	0.00	0.00
*								

Results

Interaction Curve

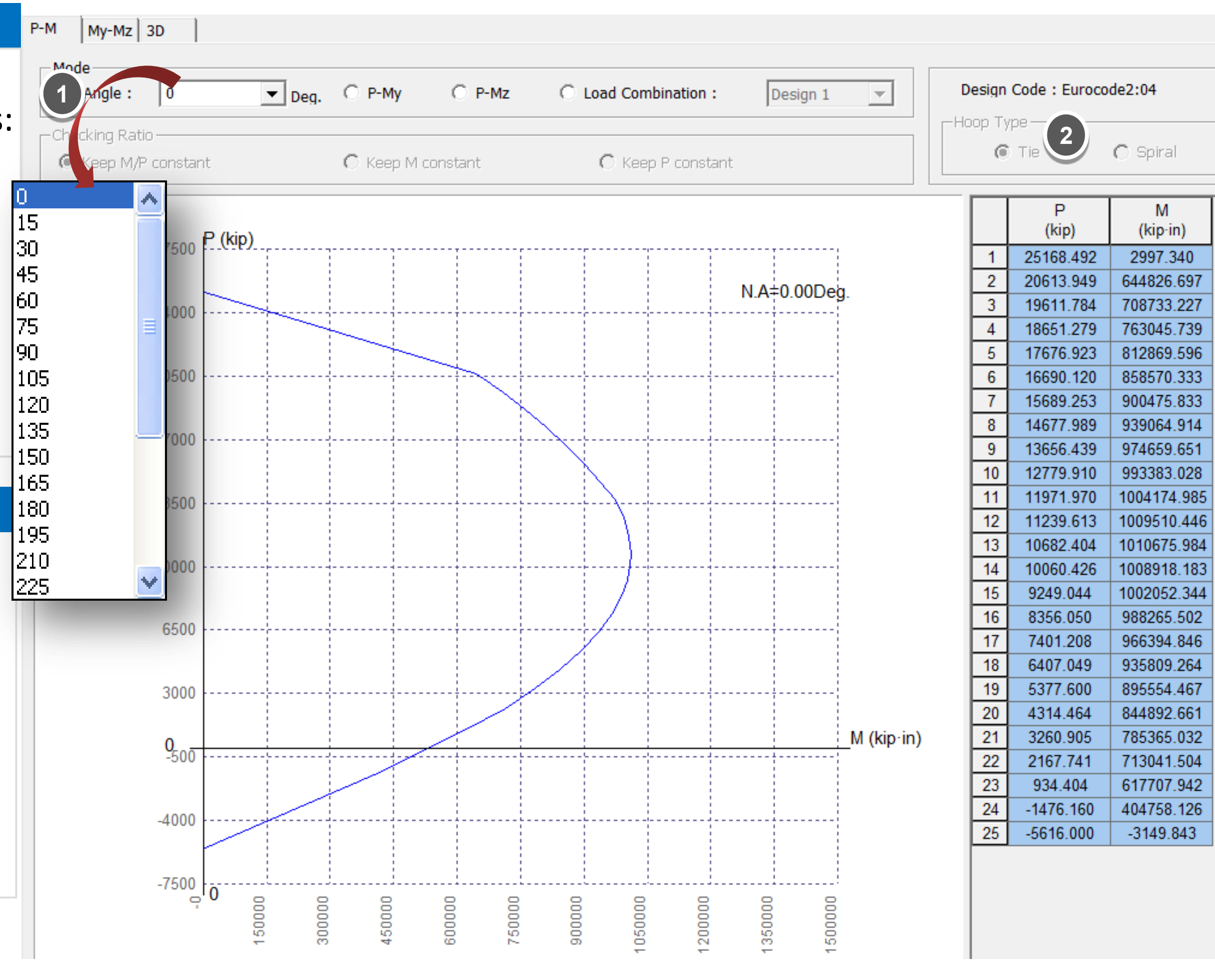
- You can see following interaction curves:
  1. P-M interaction curve.
  2. My-Mz interaction curve.
  3. 3D interaction surface.

P-M interaction curve

Result > Interaction Curve

❶ P-M curve can be seen for all the angles from 0 to 360 by 15 degrees. Select 0 from menu

❷ Select tie as type of hoop



Results

P-M interaction curve

Result > Interaction Curve

❶Select Load Combination

load combinations.

❷Select Design 1

Checking Ratio can be calculated in

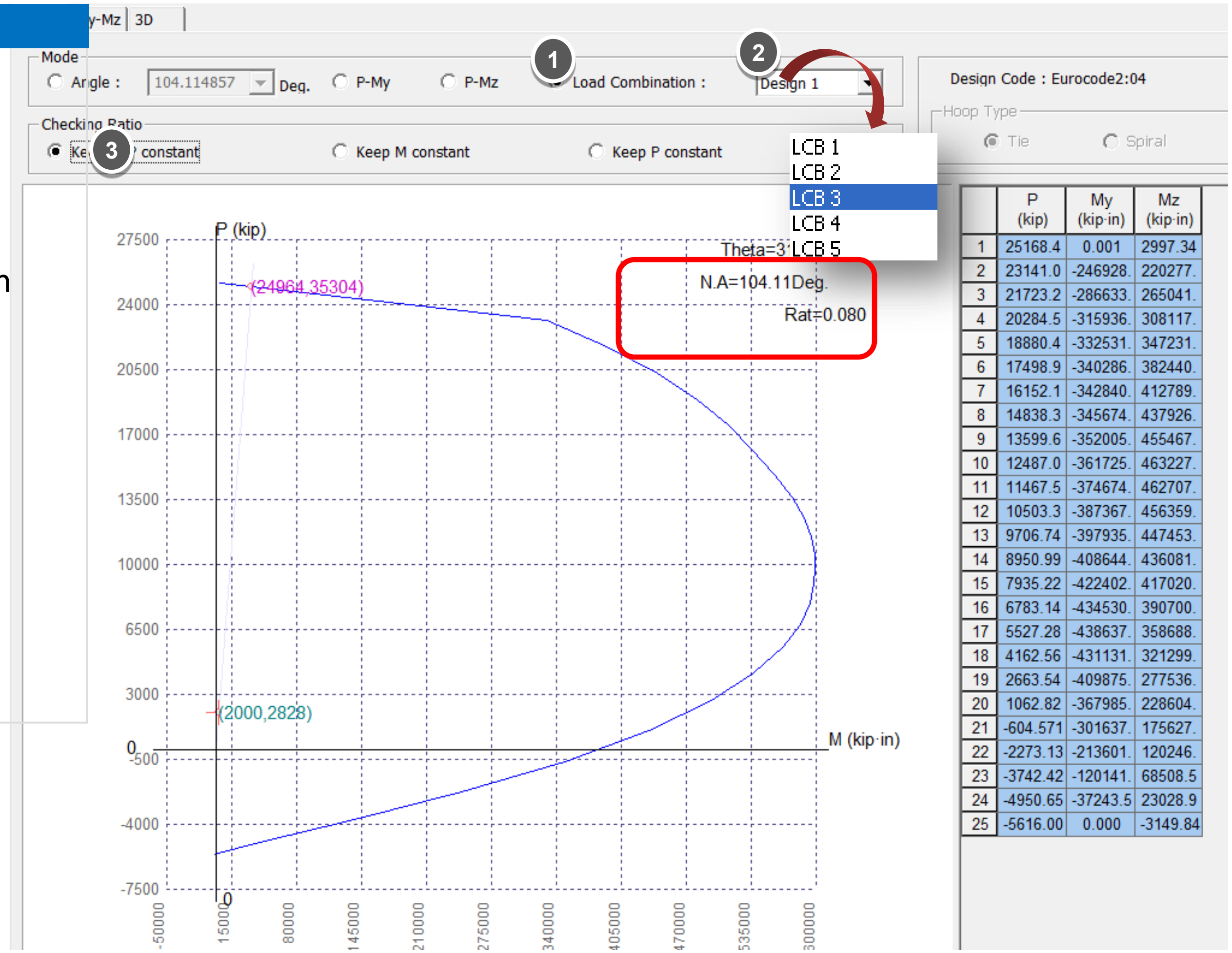
❸three ways:

1.Keep M/P constant

2. Keep M constant

3. Keep P constant

Select Keep M/P constant



	P (kip)	My (kip·in)	Mz (kip·in)
1	25168.4	0.001	2997.34
2	23141.0	-246928.	220277.
3	21723.2	-286633.	265041.
4	20284.5	-315936.	308117.
5	18880.4	-332531.	347231.
6	17498.9	-340286.	382440.
7	16152.1	-342840.	412789.
8	14838.3	-345674.	437926.
9	13599.6	-352005.	455467.
10	12487.0	-361725.	463227.
11	11467.5	-374674.	462707.
12	10503.3	-387367.	456359.
13	9706.74	-397935.	447453.
14	8950.99	-408644.	436081.
15	7935.22	-422402.	417020.
16	6783.14	-434530.	390700.
17	5527.28	-438637.	358688.
18	4162.56	-431131.	321299.
19	2663.54	-409875.	277536.
20	1062.82	-367985.	228604.
21	-604.571	-301637.	175627.
22	-2273.13	-213601.	120246.
23	-3742.42	-120141.	68508.5
24	-4950.65	-37243.5	23028.9
25	-5616.00	0.000	-3149.84

Results

3-D interaction surface

Result > Interaction Curve

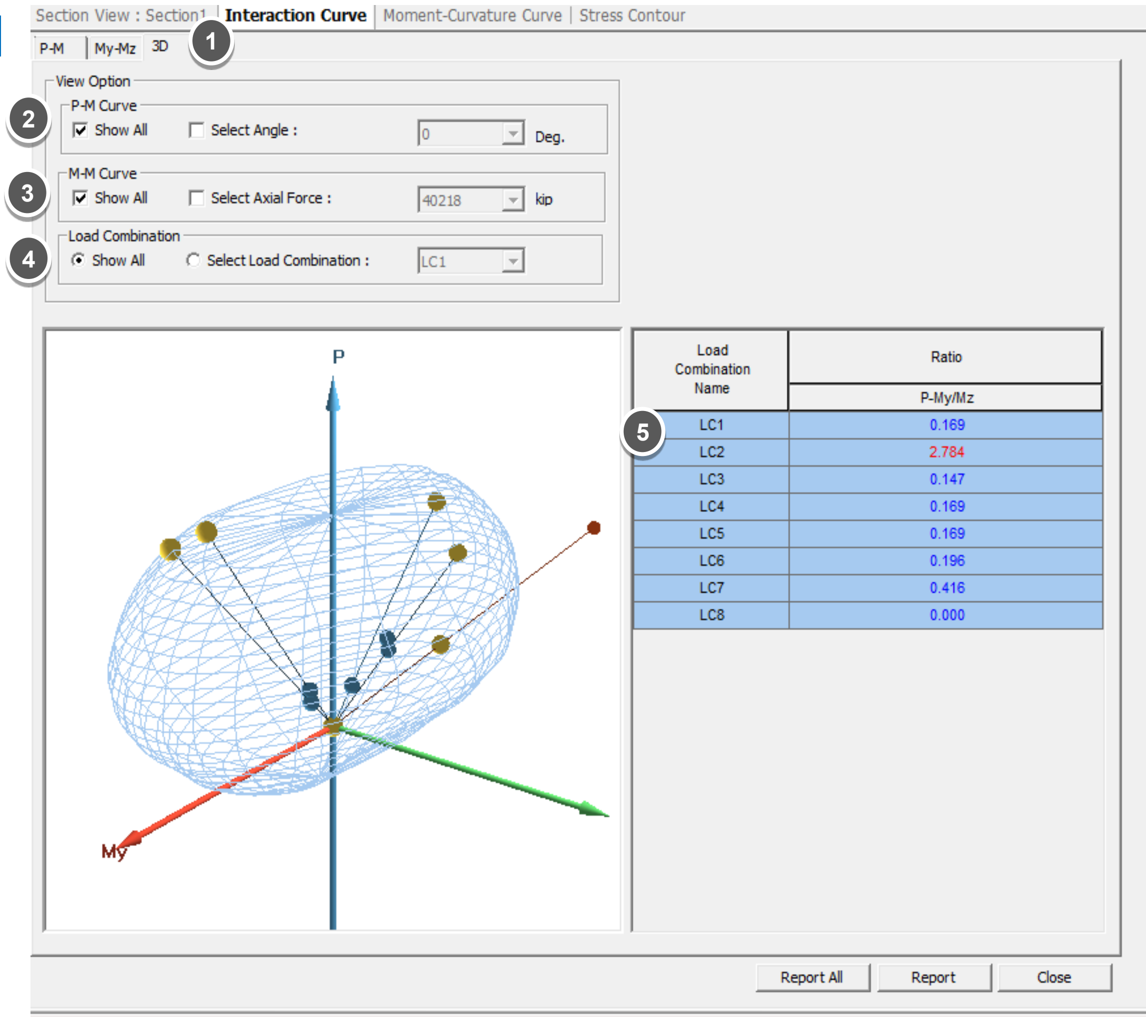
❶ Click on “**3D**” tab

❷ Select Show All in View Option to see all the P-M curves at the same time.

❸ Select Show All in M-M Curve

❹Select Show All in Load Combination

❺The Capacity Ratio is shown for all the load combinations. The ratio > 1 is shown in red (LC2)



Results

Moment –Curvature curve

Result > Moment –Curvature Curve

or

❶ Click on “**Moment-Curvature**” tab.

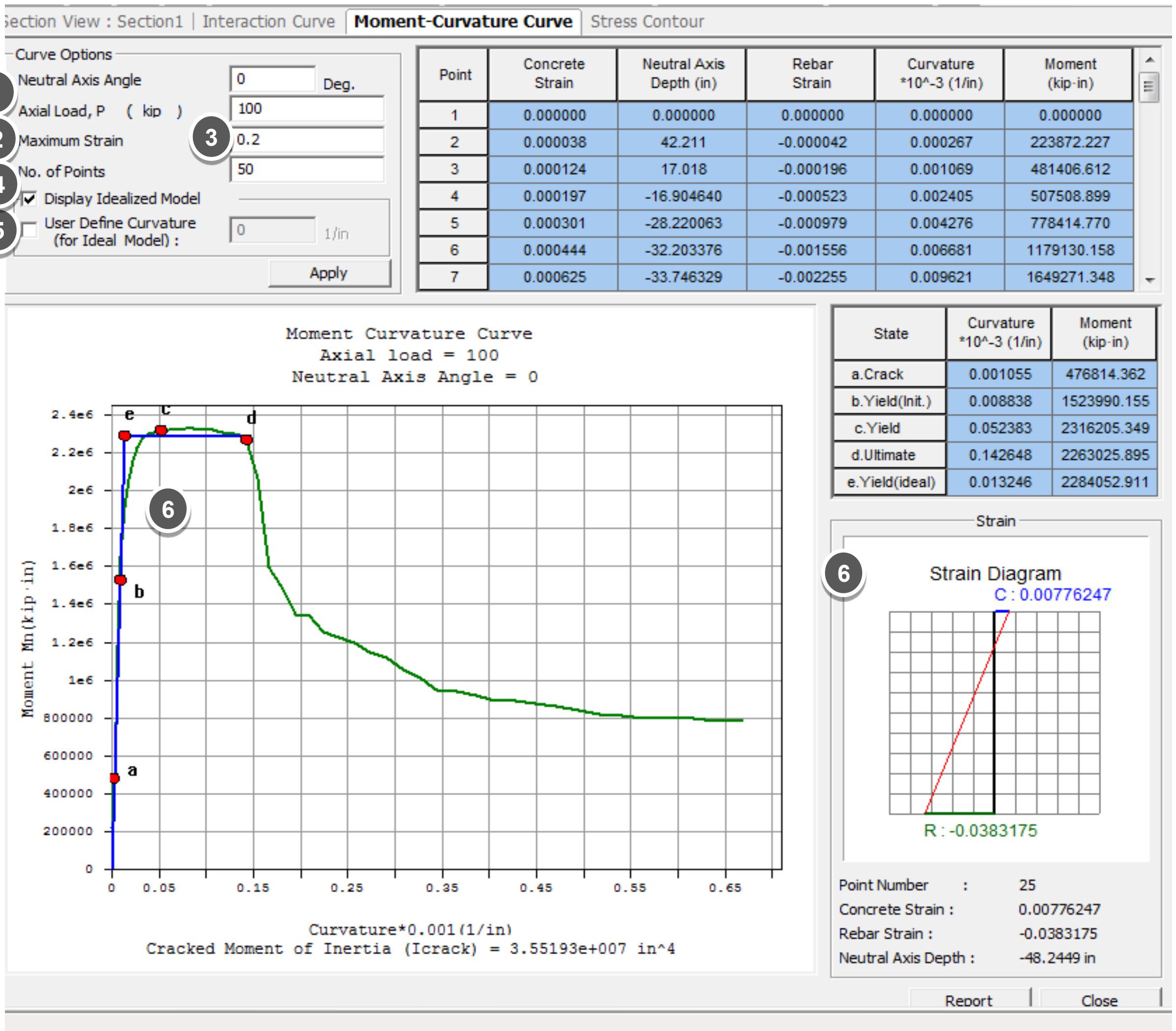
❷ Enter 100 kips as Axial Load P

❸ Enter 0.2 as the Maximum Strain

❹ Number of Points: 50

❺ Display Idealized Model → Apply

❻ Click on different points on curve to see strain diagram for a each point



Results

Stress Contours

Result > Stress Contours

OR

Click on "**Stress Contours**" tab.

❶ Select the Load Combination LCB4

❷ Check Stress for both Concrete and Rebar

❸ Check the Max/Min & N.A value

Sign Convention

- Tension is taken as +ve and compression as –ve.