

Pier Modeling Example

- *Bridging Your Innovations to Realities*

Training material



midas **Civil**



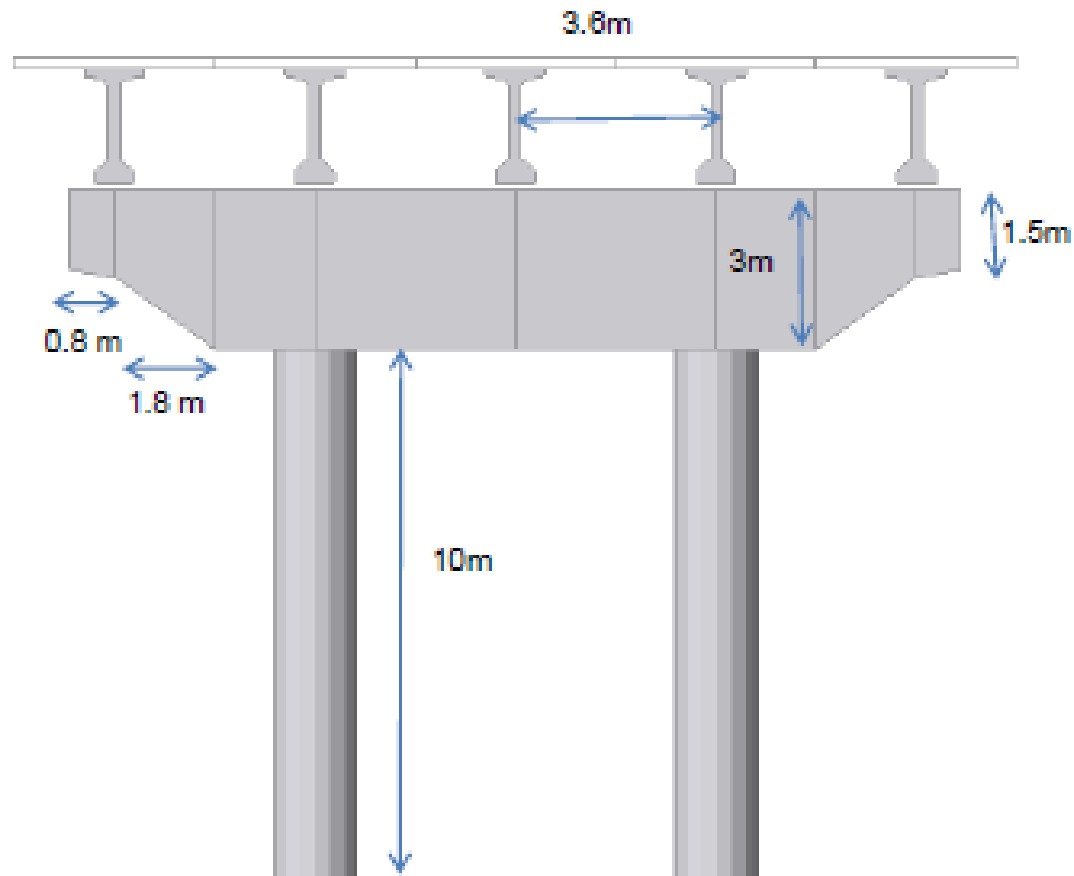
Overview: Pier Modeling

1. Response Spectrum Analysis

1. Introduction
2. Material
3. Sections
4. Modeling



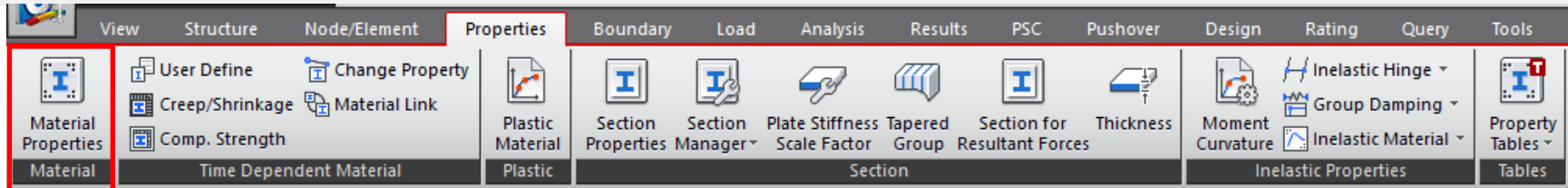
1. Introduction: Pier Modeling



Front View



2. Material



Material Data

General
Material ID: 1 Name: Grade C5000

Elasticity Data
Type of Design: Concrete (1)

Type of Material
☒ Isotropic ☐ Orthotropic

Steel
Standard: DB
Concrete (2)
Standard: ASTM(RC)
Code: DB
Grade C5000 (3)

Steel
Modulus of Elasticity: 0.0000e+000 tonf/m²
Poisson's Ratio: 0
Thermal Coefficient: 0.0000e+000 1/[F]
Weight Density: 0 tonf/m³
☐ Use Mass Density: 0 tonf/m³/g

Concrete
Modulus of Elasticity: 2.8645e+006 tonf/m²
Poisson's Ratio: 0.2
Thermal Coefficient: 5.0000e-006 1/[F]
Weight Density: 2.403 tonf/m³
☐ Use Mass Density: 0.245 tonf/m³/g

Plasticity Data
Plastic Material Name: NONE

Thermal Transfer
Specific Heat: 0 Btu/tonf*[F]
Heat Conduction: 0 Btu/m*hr*[F]

Damping Ratio: 0.05

OK Cancel Apply

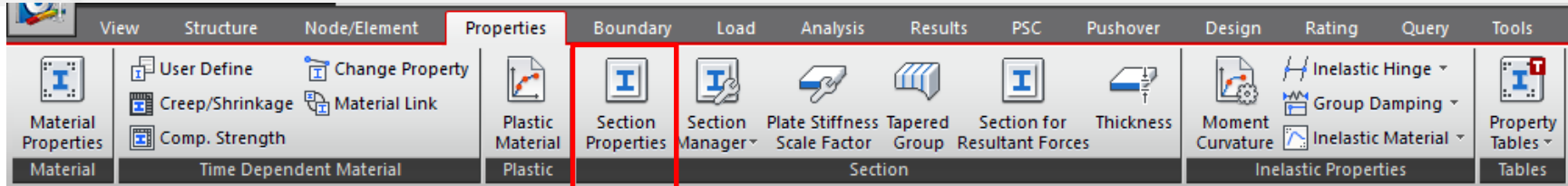
Properties > Material Properties

Concrete:

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



3. Sections



Section Data

DB/User Value SRC Combined PSC Tapered Composite Steel Girder

Section ID: 1

Name: Cap End

Section-i

i-Name: Solid Rectangle

H: 1.5 m B: 1.5 m

Section-j

j-Name:

H: 3 m B: 1.5 m

y Axis Variation: Linear

z Axis Variation: Linear

☒ Consider Shear Deformation.

☐ Consider Warping Effect(7th DOF)

Offset: Center-Top

Change Offset ...

Display Centroid

Show Calculation Results... OK Cancel Apply

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:

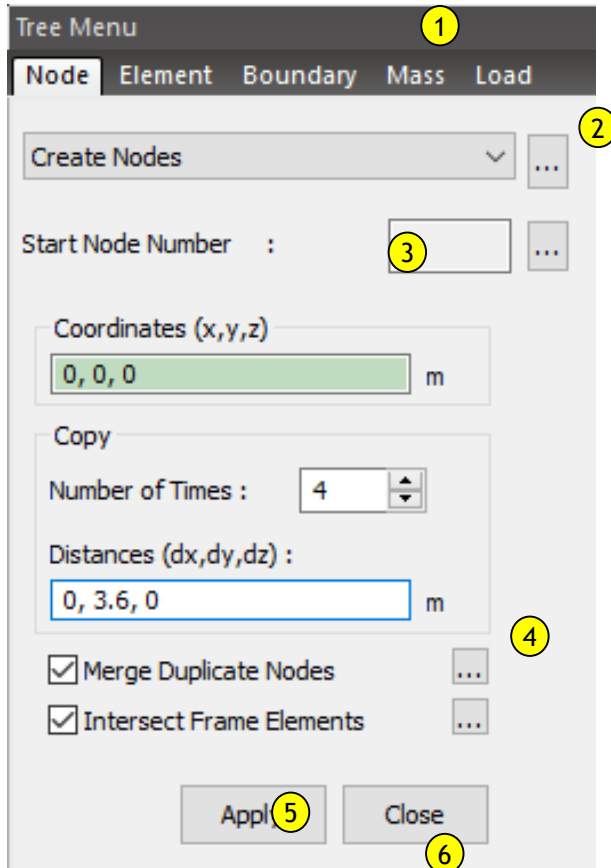
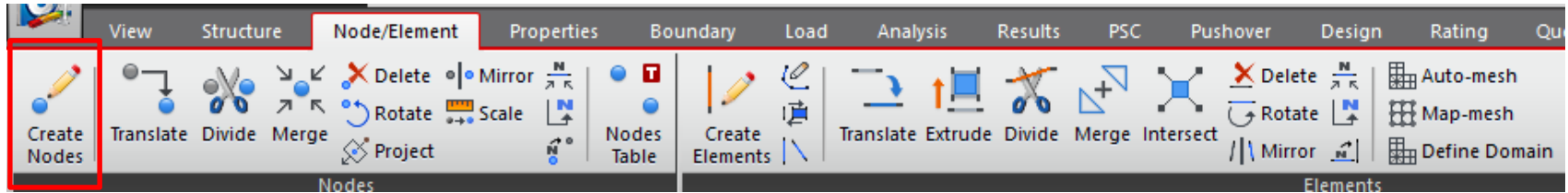
1. Select DB/USER tab
2. Select Solid Rectangle
3. Enter 3 m X 1.5 m
4. Set Offset at Center Top
5. Click Apply

Pier:

1. Select Solid Round
2. Enter diameter as 1.5 m
3. Set Offset at Center - Center
4. Click OK



4. Modeling



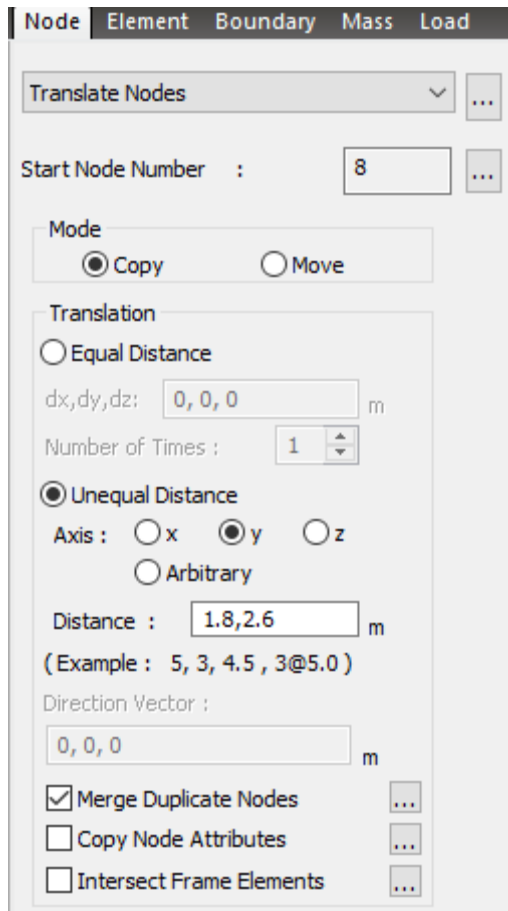
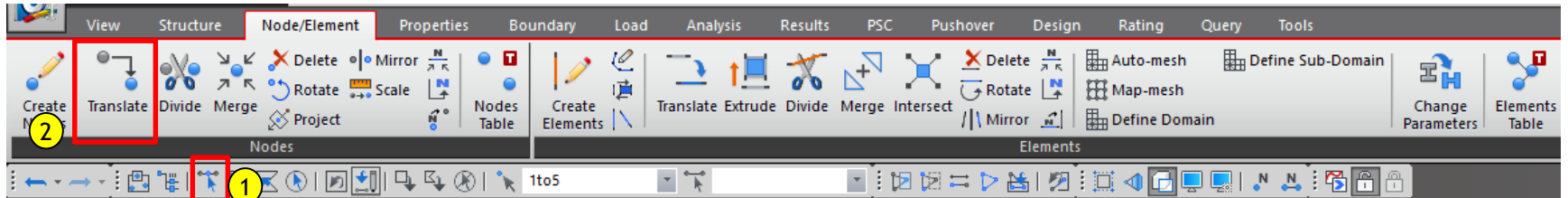
Node/Element > Create Nodes

Add Section

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

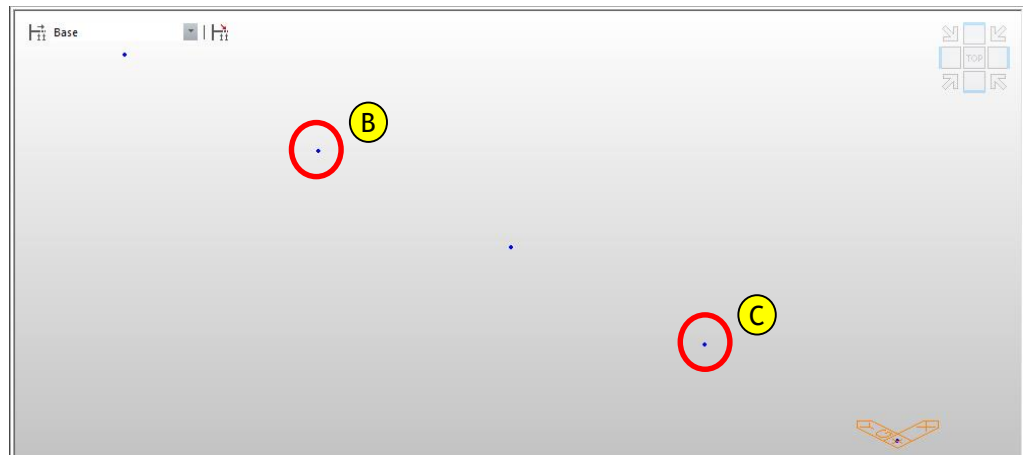


4. Modeling



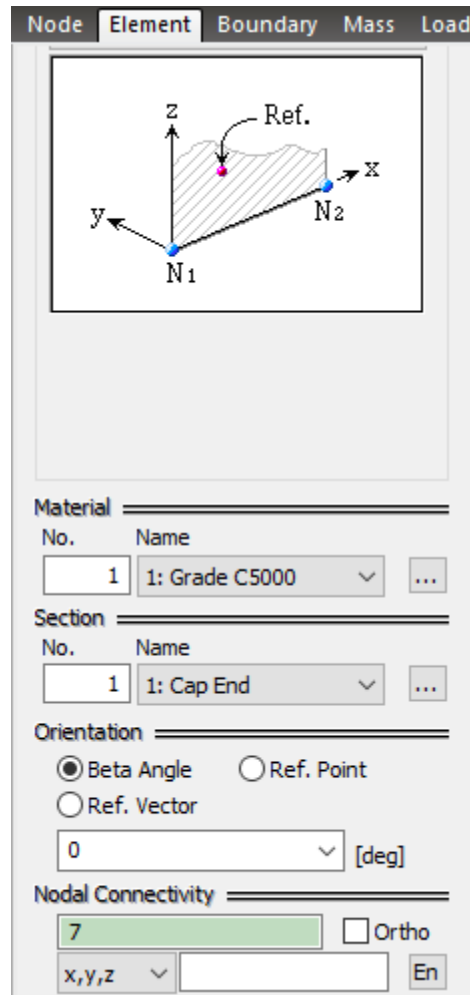
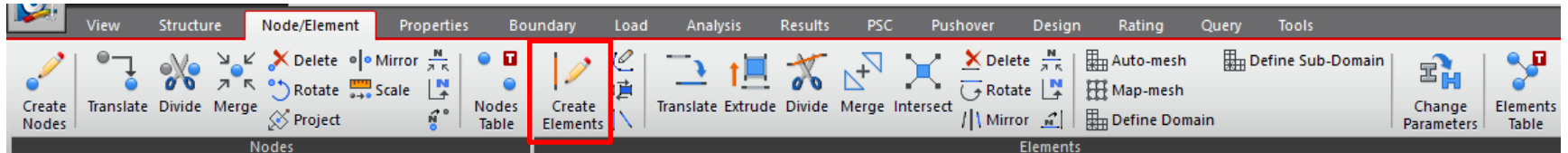
Node/Element > Translate Nodes

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



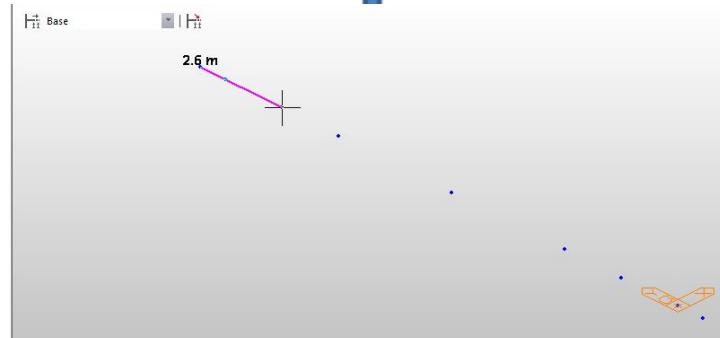
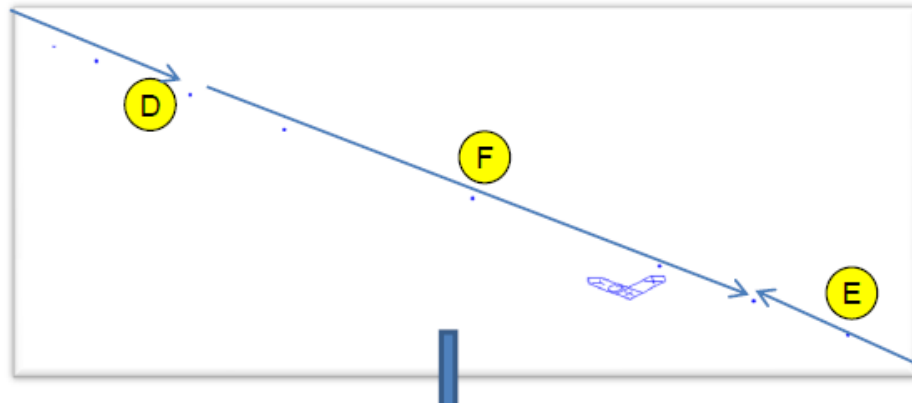


4. Modeling



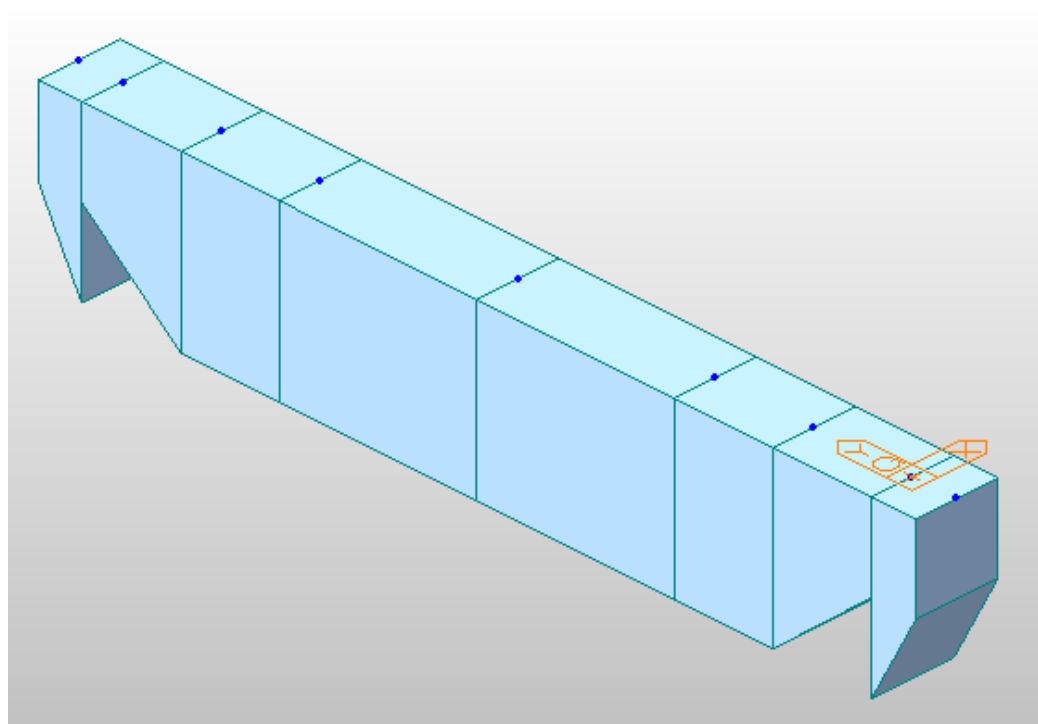
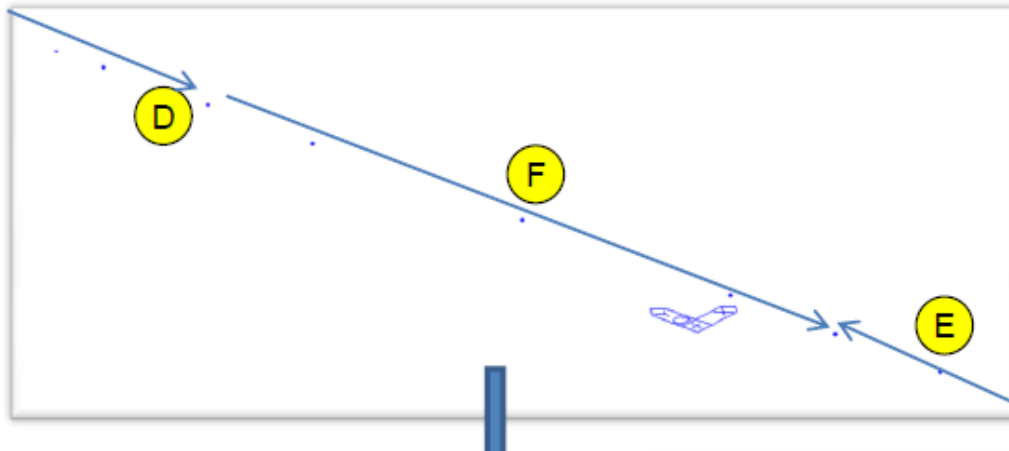
Node/Element > Create Elements

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]



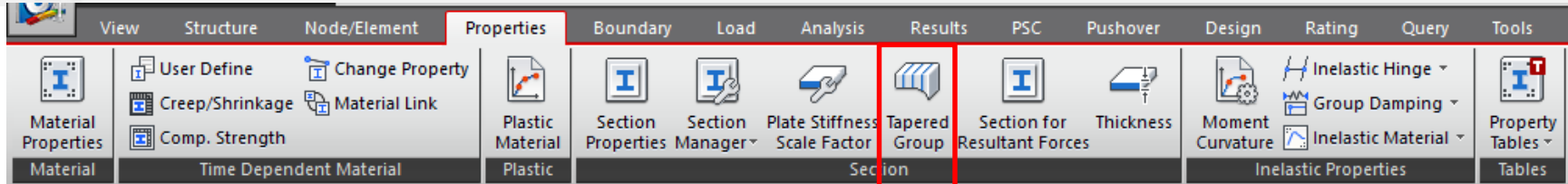


4. Modeling



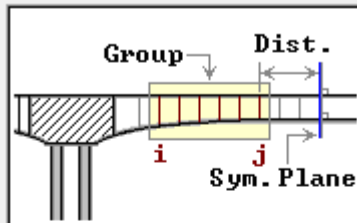


4. Modeling



Node Element Boundary Mass Load

Tapered Section Group



Group Name : Left_End

Element List :

1 2

Section Shape Variation

z-Axis

☐ Linear ☒ Polynomial 2.0

Symmetric Plane

From : ☒ i ☐ j

Distance : 0 m

y-Axis

☒ Linear ☐ Polynomial 2.0

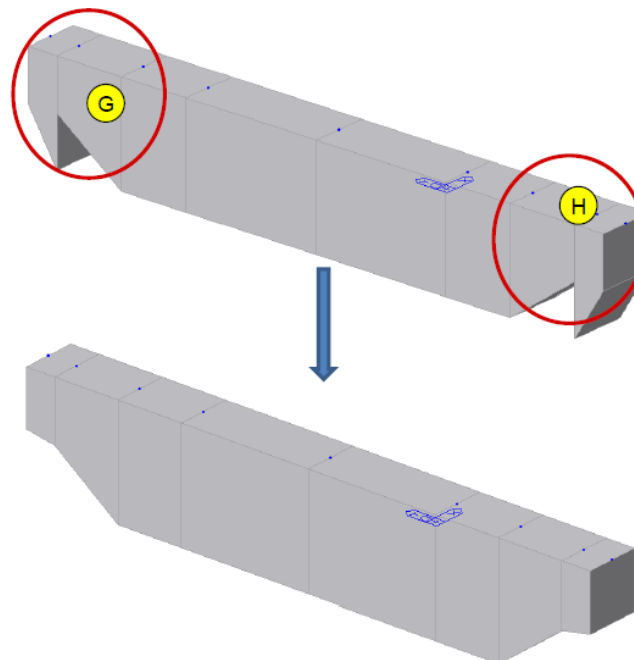
Symmetric Plane

From : ☐ i ☐ j

Distance : 0 m

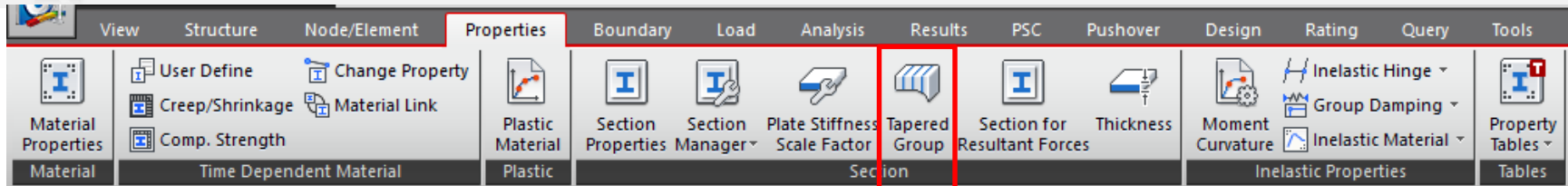
Properties > Tapered Group

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)



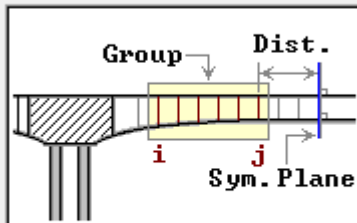


4. Modeling



Node Element Boundary Mass Load

Tapered Section Group



Group Name : Right_End

Element List :

3 4

Section Shape Variation

z-Axis

☐ Linear ☒ Polynomial 2.0

Symmetric Plane

From : ☒ i ☐ j

Distance : 0 m

y-Axis

☒ Linear ☐ Polynomial 2.0

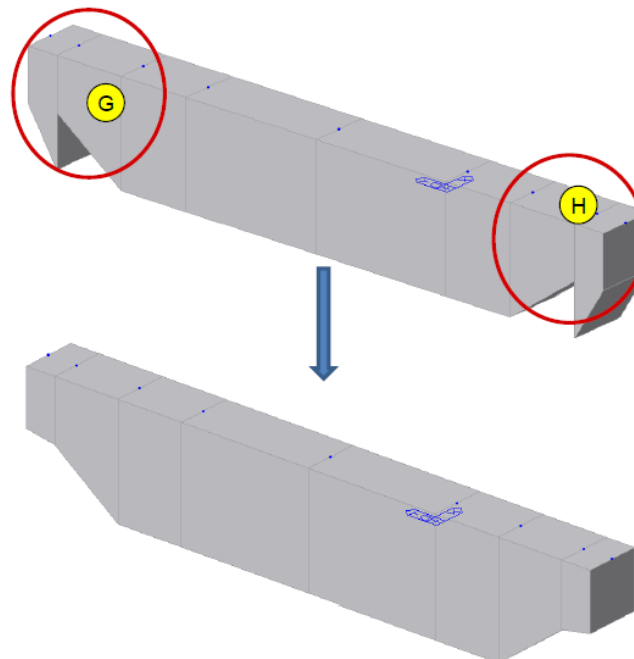
Symmetric Plane

From : ☐ i ☐ j

Distance : 0 m

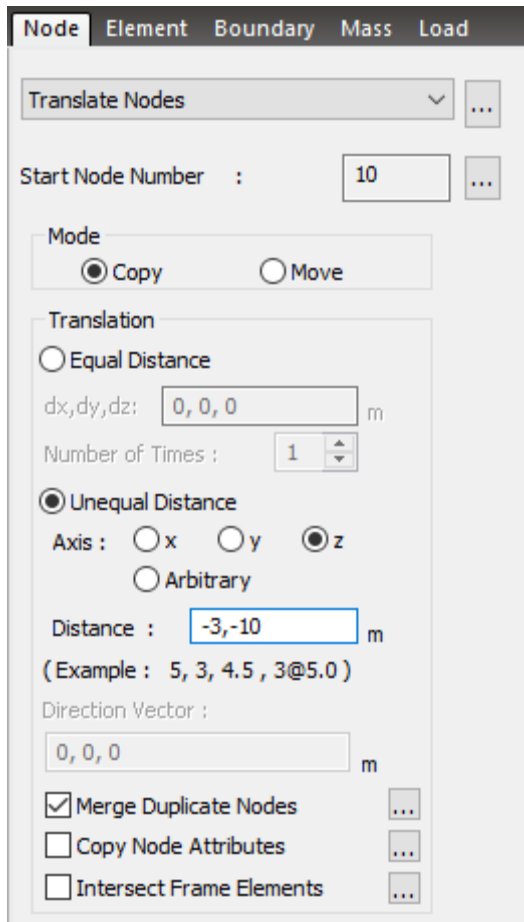
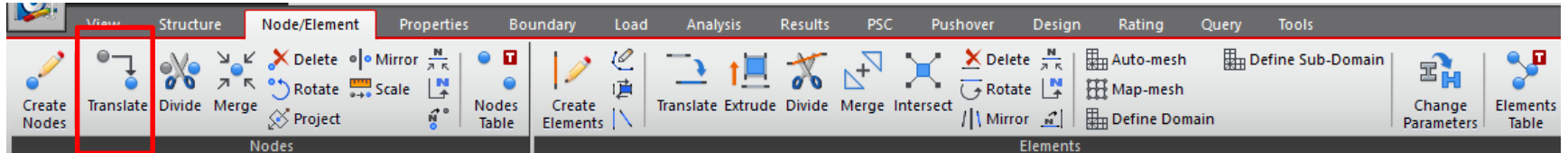
Properties > Tapered Group

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)



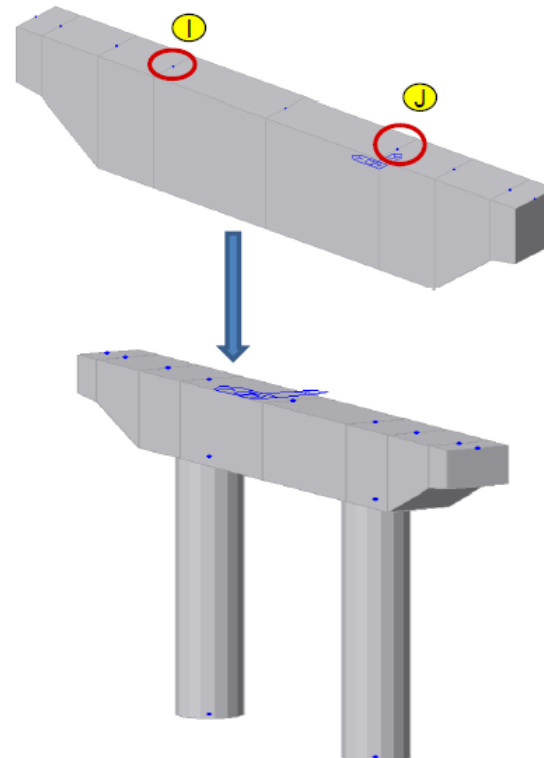


4. Modeling



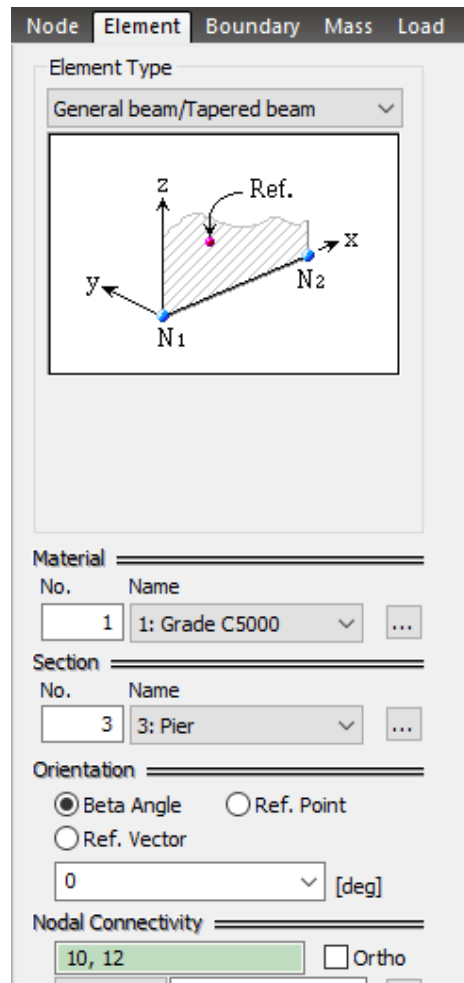
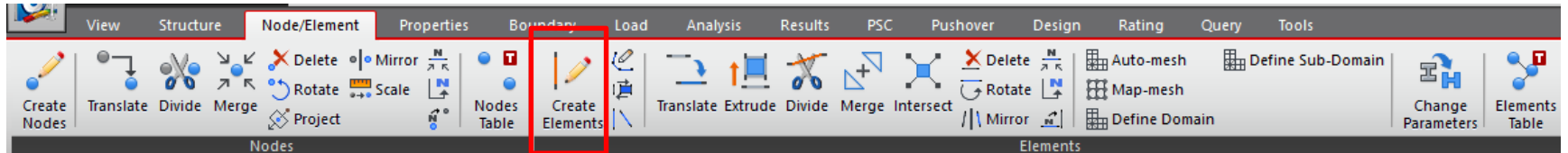
Node/Element>Translate

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



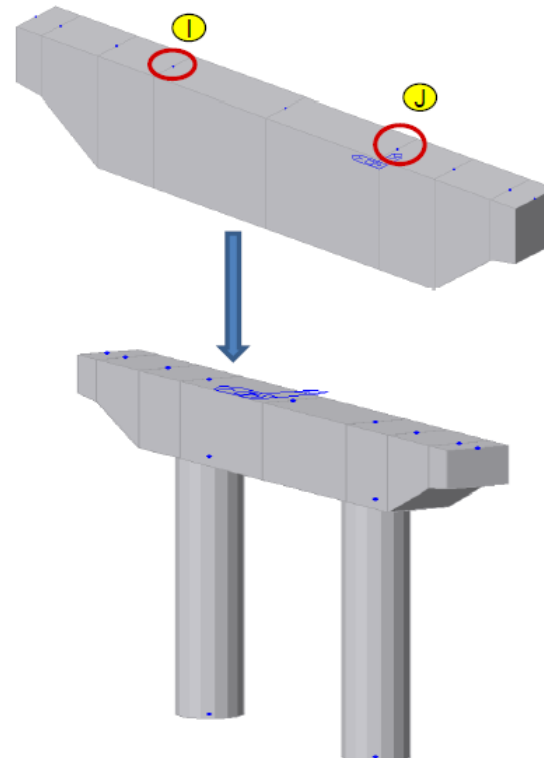


4. Modeling



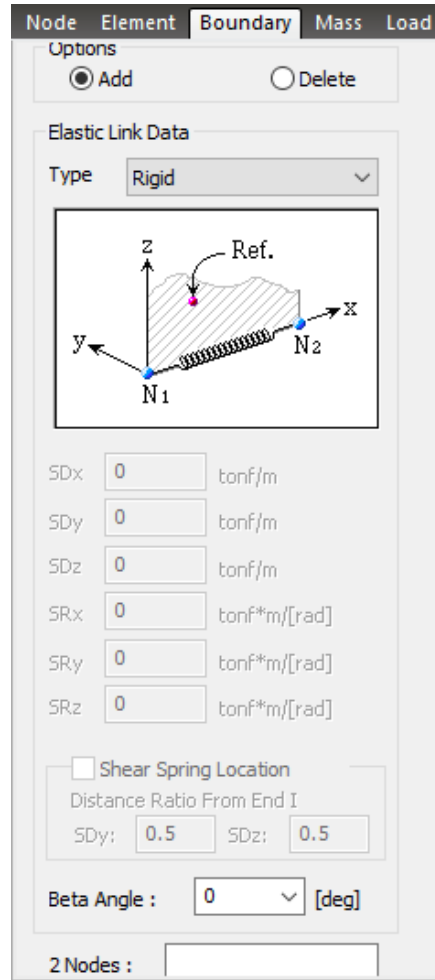
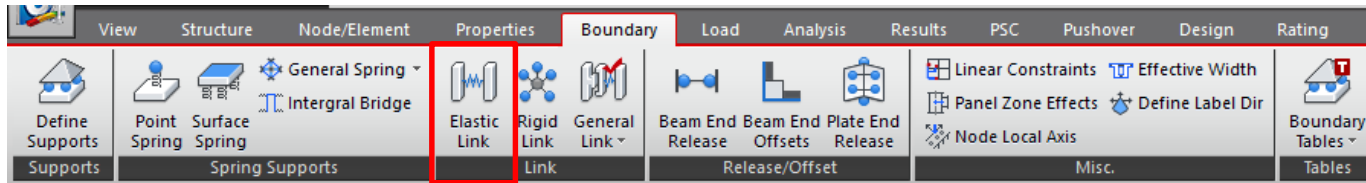
Node/Element>Translate

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



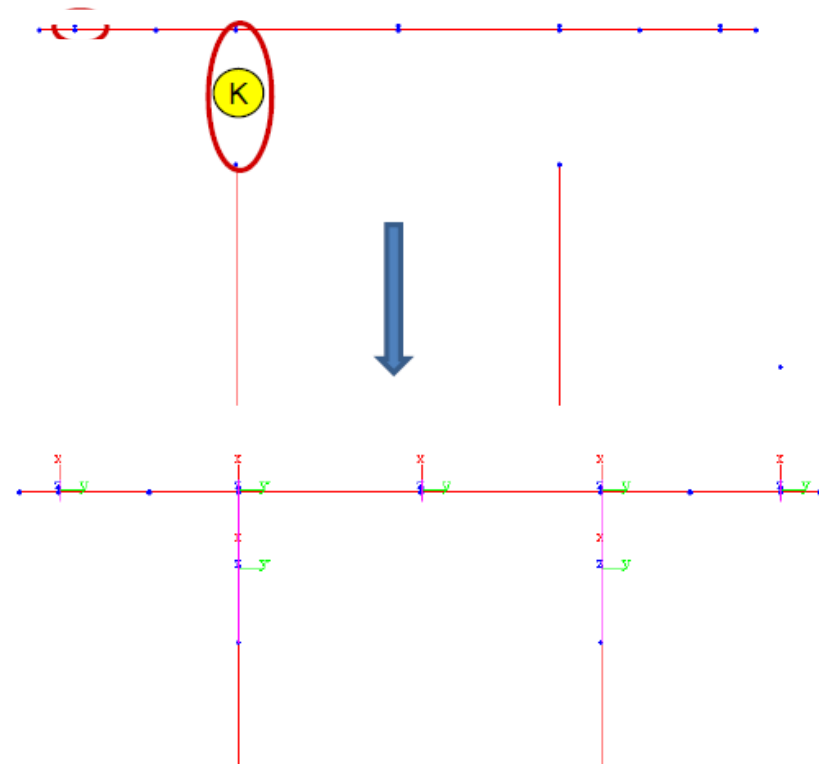


4. Modeling



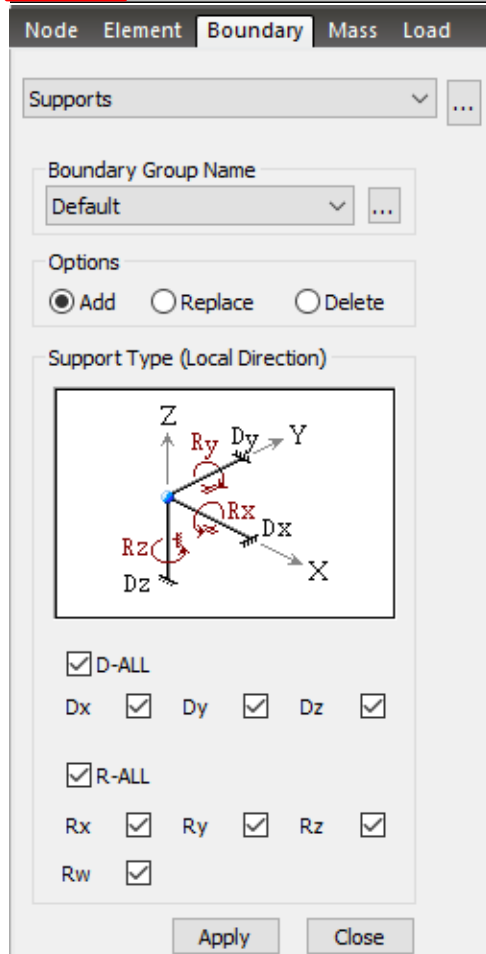
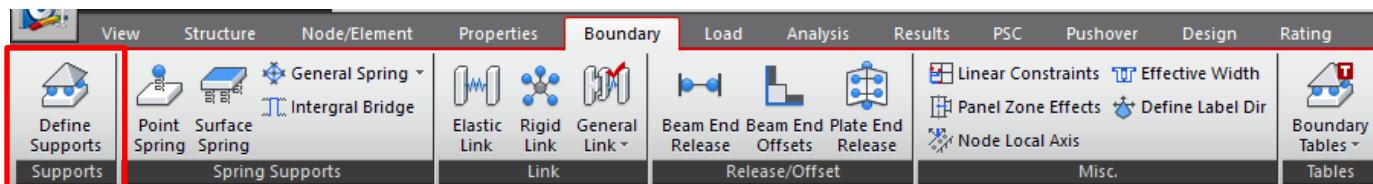
Boundary > Elastic Link

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



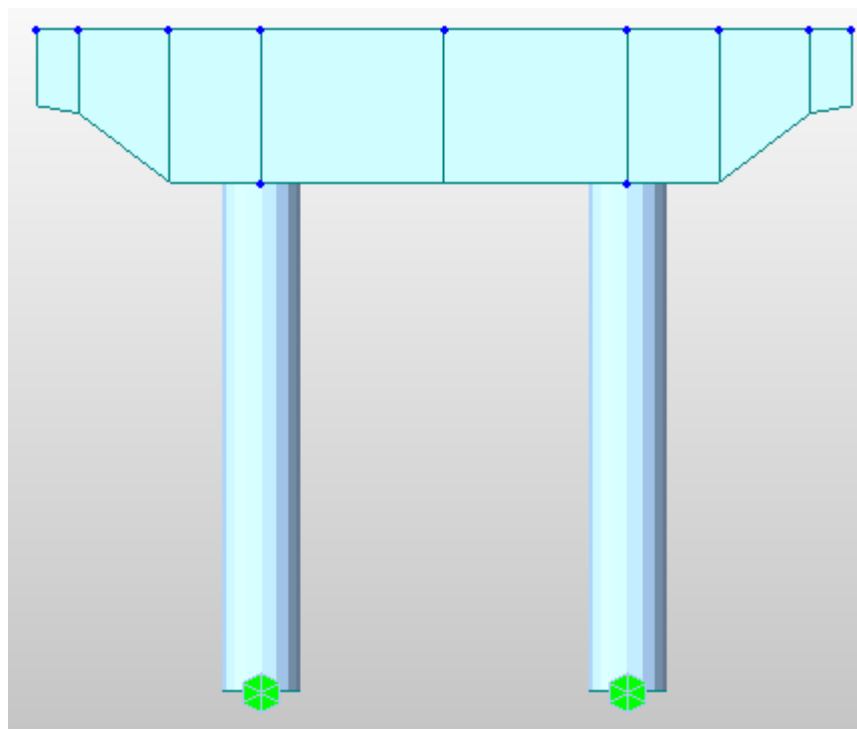


4. Modeling



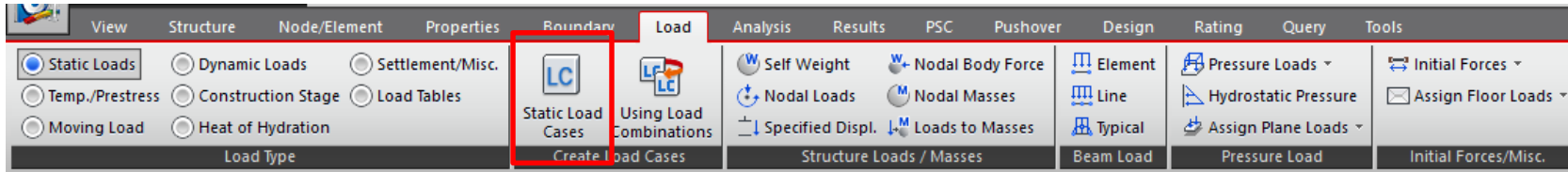
Boundary > Define Supports

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

Static Load Cases



Static Load Cases dialog box showing the configuration for a new load case.

Name :

Case :

Type :

Description :

No	Name	Type	Description
1	Dead Load	Dead Load (D)	



4. Modeling

Software interface showing the **Load** menu and the **Nodal Loads** dialog box.

Load Menu:

- Static Loads (selected)
- Dynamic Loads
- Settlement/Misc.
- Temp./Prestress
- Construction Stage
- Load Tables
- Moving Load
- Heat of Hydration

Load Type: Static Load Cases, Using Load Combinations, Create Load Cases

Analysis: Self Weight, Nodal Body Force, Nodal Masses, Specified Displ., Loads to Masses, Structure Loads / Masses

Design: Element, Line, Typical, Beam Load

Rating: Pressure Loads, Hydrostatic Pressure, Assign Plane Loads, Pressure Load

Tools: Initial Forces, Assign Floor Loads, Initial Forces/Misc.

Nodal Loads Dialog Box:

- Load Case Name: Dead Load
- Load Group Name: Default
- Options: Add (selected), Replace, Delete
- Nodal Loads:
- FX: 0 kips
- FY: 0 kips
- FZ: -290.5 kips
- MX: 0 kips*m
- MY: 0 kips*m
- MZ: 0 kips*m

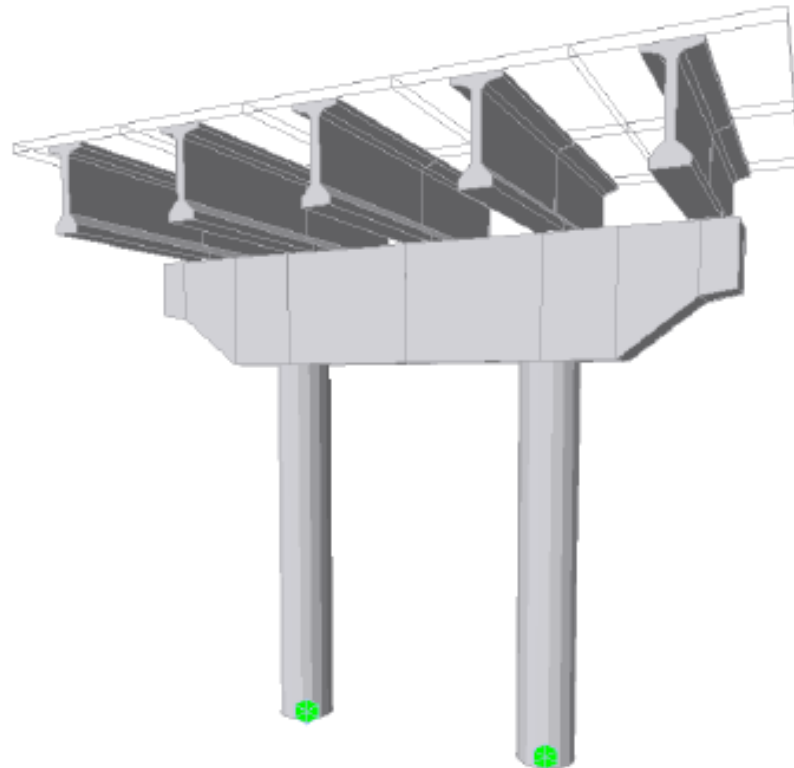
Load > Static Loads > Nodal Loads

- Select the nodes shown to apply the dead load as nodal loads
- Apply -290.5 kips in the FZ (Vertical direction)

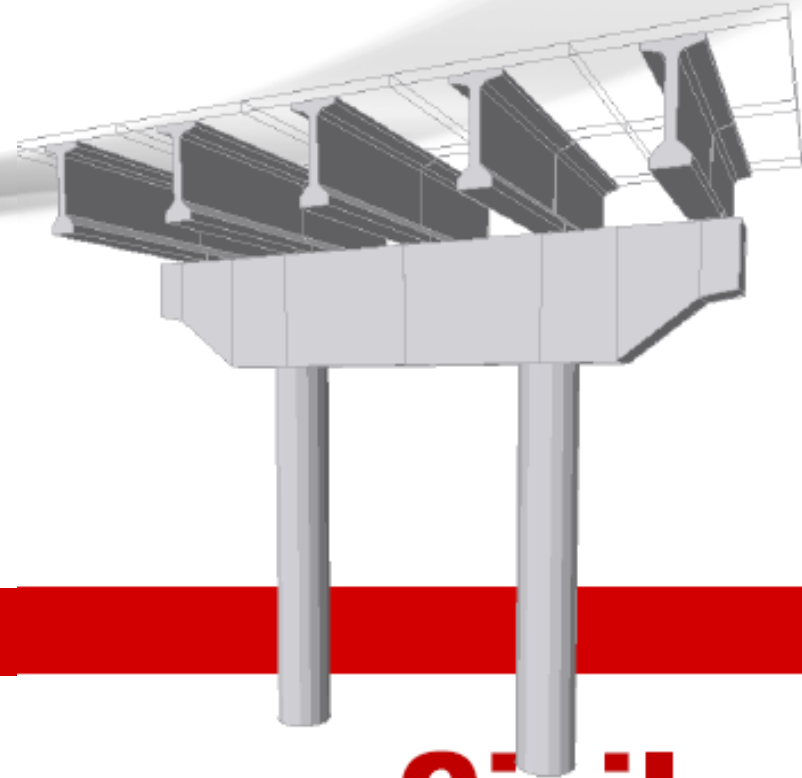


4. Modeling

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*

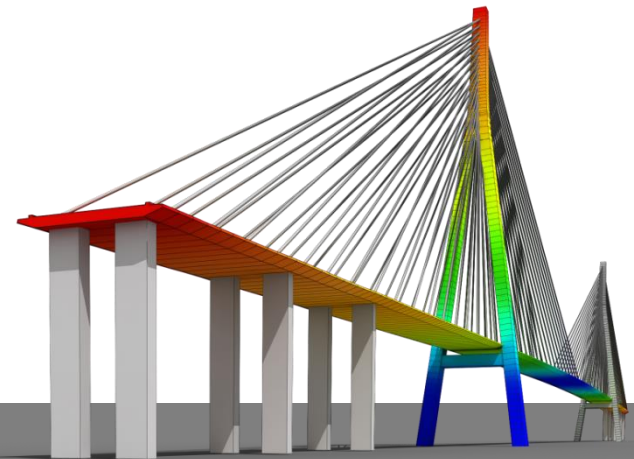
Training material

midas **Civil**

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

Section Design as per AASHTO-LRFD 2012 using midas GSD

- *Bridging Your Innovations to Realities*

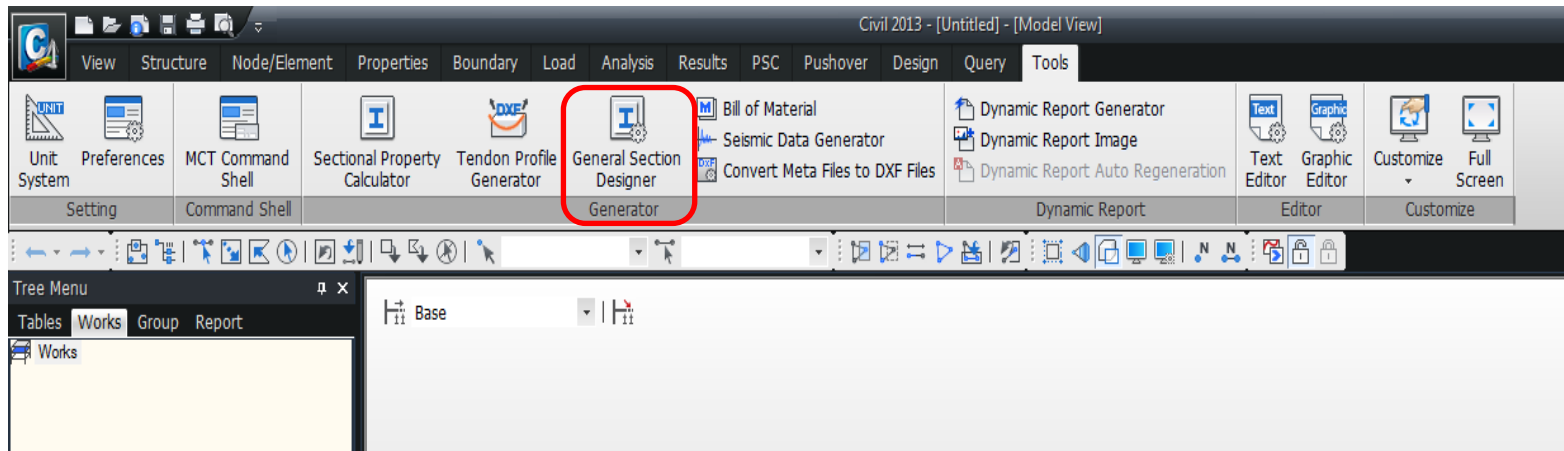


► Introduction

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.

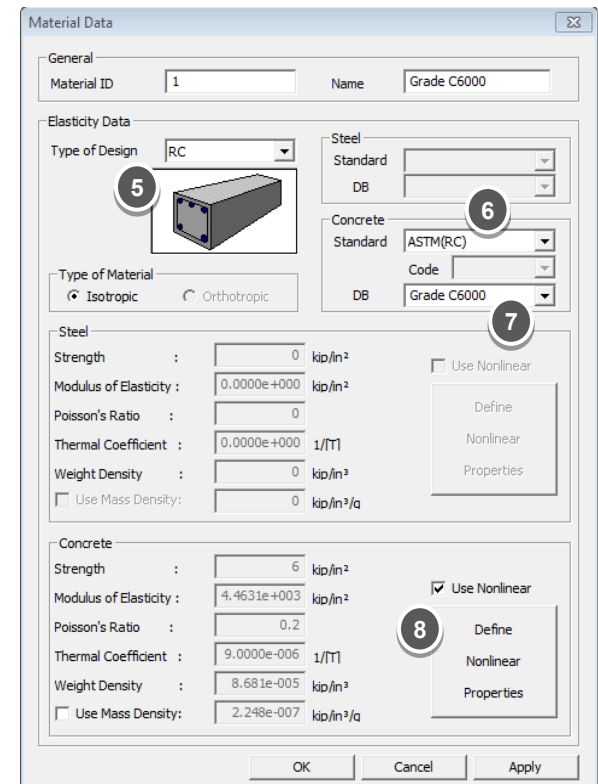
► Start

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

Nonlinear properties

Hysteresis Model (Curve Type)
Mander Model

Compressive Stress, f_c

Compressive Strain, ϵ_c

Unconfined Concrete

Confined Concrete

Assumed for Cover Concrete

Elastic modulus of Concrete : $E_c = 5,000\sqrt{f'_{co}}$ MPa

Tensile Strength of Concrete : $f'_t = 0.62\sqrt{f'_{co}}$ MPa

Tensile Strain of Concrete : $\epsilon_t = \frac{f'_t}{E_c}$

Skeleton Curve

f'_{co} : 6 kip/in²

ϵ_{co} : 0.002

E_c : Mander

f'_t : Mander

ϵ_t : 1.240e-004


ϵ_{sp} : 0.005

4664.296 kip/in²

0.578 kip/in²

OK Cancel

► 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

Basic Shape Data

Shape ID: 1

Name: B5181.6x2133.6x3

Box ☒ User ☐ DB ☐ UNI

Shape Name:

☒ Built-Up Section

Get Data from Single Angle

DB Name: KS

Shape Name:

H: 204 in

B: 84 in

t_w : 15 in

t_{f1} : 16 in

C: 0 in

t_{f2} : 0 in

Material: Grade C6000

Insertion Point: 42, 102

Rotation Angle: 0 degree

☐ Hollow Shape

OK Cancel Apply

► Reinforcement

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

The screenshot shows the 'Rebar Material Property' dialog box. It contains the following fields and controls:

- Rebar Material Code:** A dropdown menu set to 'ASTM(RC)', with a circled '4' above it.
- Rebar Grade:** A dropdown menu set to 'Grade 60', with a circled '5' above it.
- Rebar Fy:** A text input field containing '60', with the unit 'kip/in²' to its right.
- Modulus of Elasticity:** A text input field containing '29000', with the unit 'kip/in²' to its right.
- Stress Strain Curve:** A dropdown menu set to 'Bilinear Model', with a circled '6' above it and an ellipsis button to its right.
- Buttons:** 'OK' and 'Cancel' buttons at the bottom, with a circled '7' above the 'OK' button.

► 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

Rebar Properties - Perimeter pattern

Rebar Material: Grade 60

Rebar Pattern: Perimeter 1

Rebar Dia: #11


Distance between concrete face to center of rebar: 3.3 in

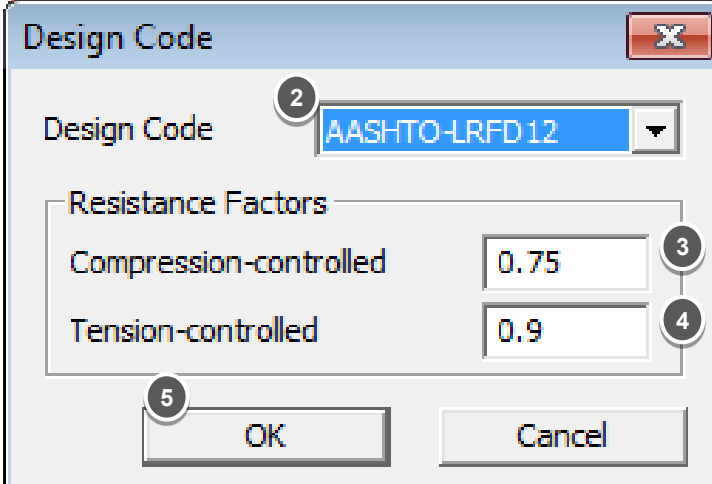
No. of Rebars: 69

Direction: ☒ Inner ☐ Outer

Buttons: OK, Cancel, 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



Design Code

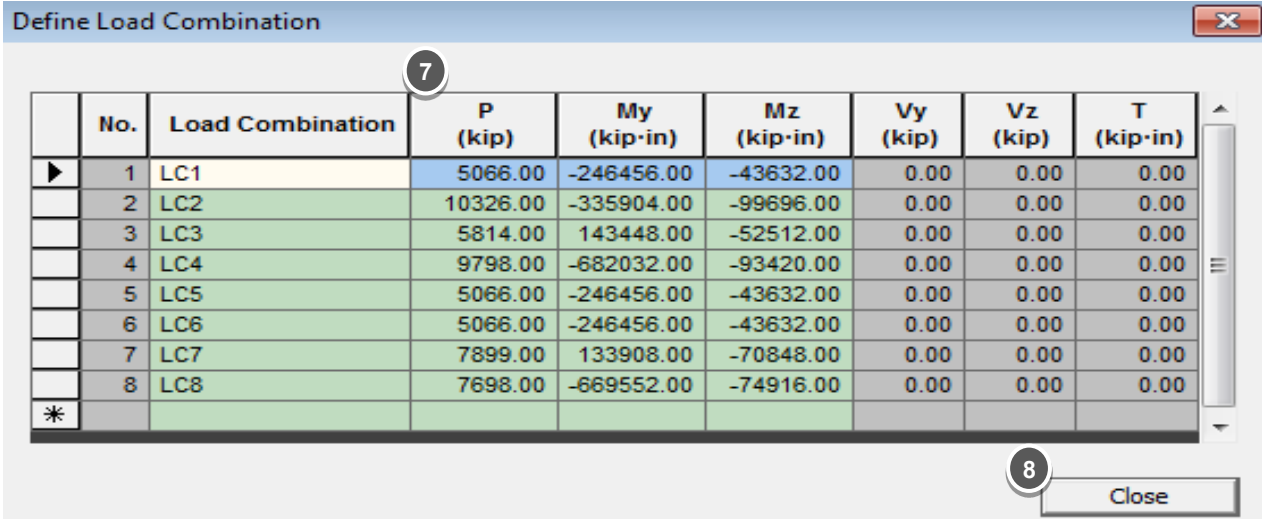
Design Code: AASHTO-LRFD 12

Resistance Factors

Compression-controlled: 0.75

Tension-controlled: 0.9

OK Cancel



Define Load Combination

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
*							

Close

Results

Interaction Curve

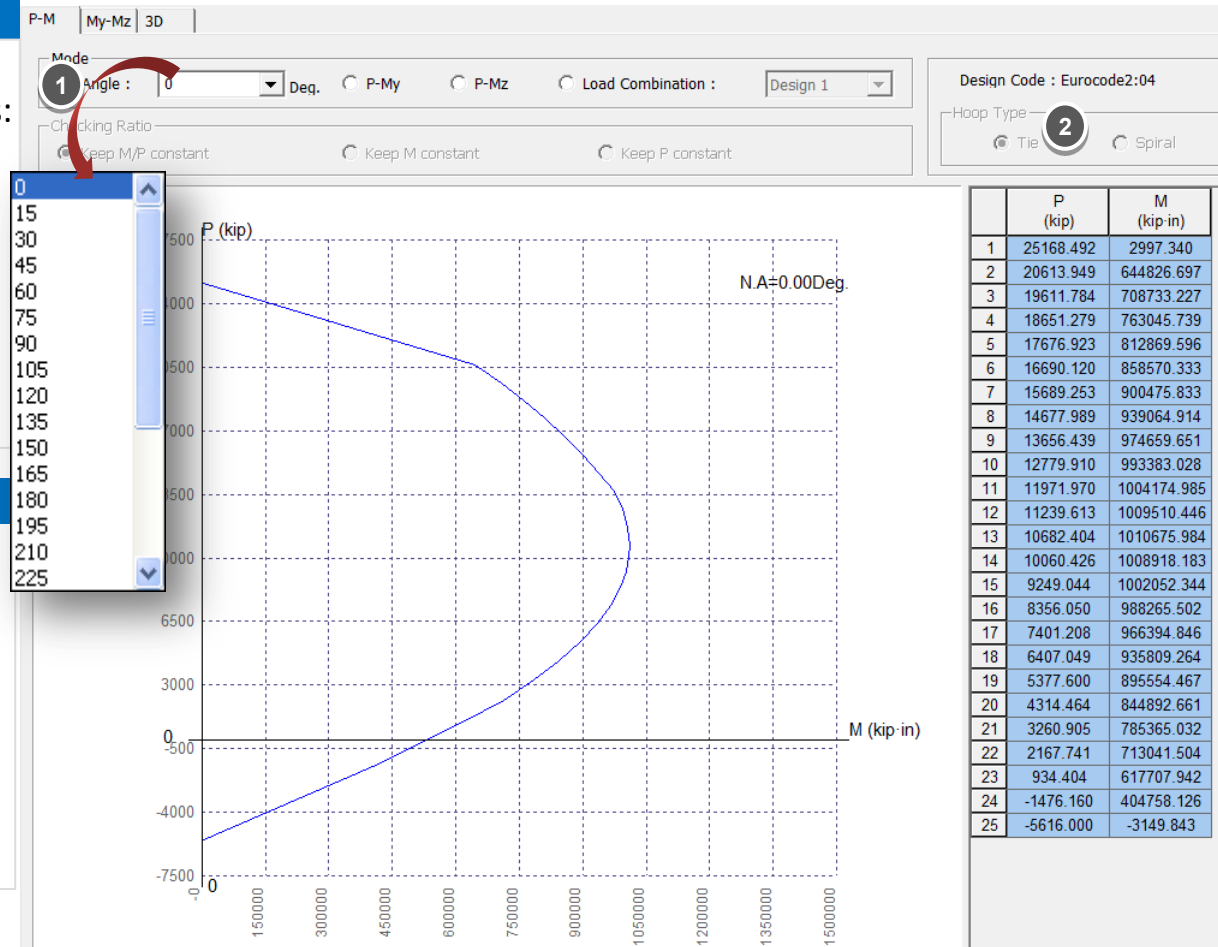
- You can see following interaction curves:

1. P-M interaction curve.
2. My-Mz interaction curve.
3. 3D interaction surface.

P-M interaction curve

Result > Interaction Curve

- ① P-M curve can be seen for all the angles from 0 to 360 by 15 degrees. Select 0 from menu
- ② Select tie as type of hoop



Results

P-M interaction curve

Result > Interaction Curve

① Select Load Combination
load combinations.

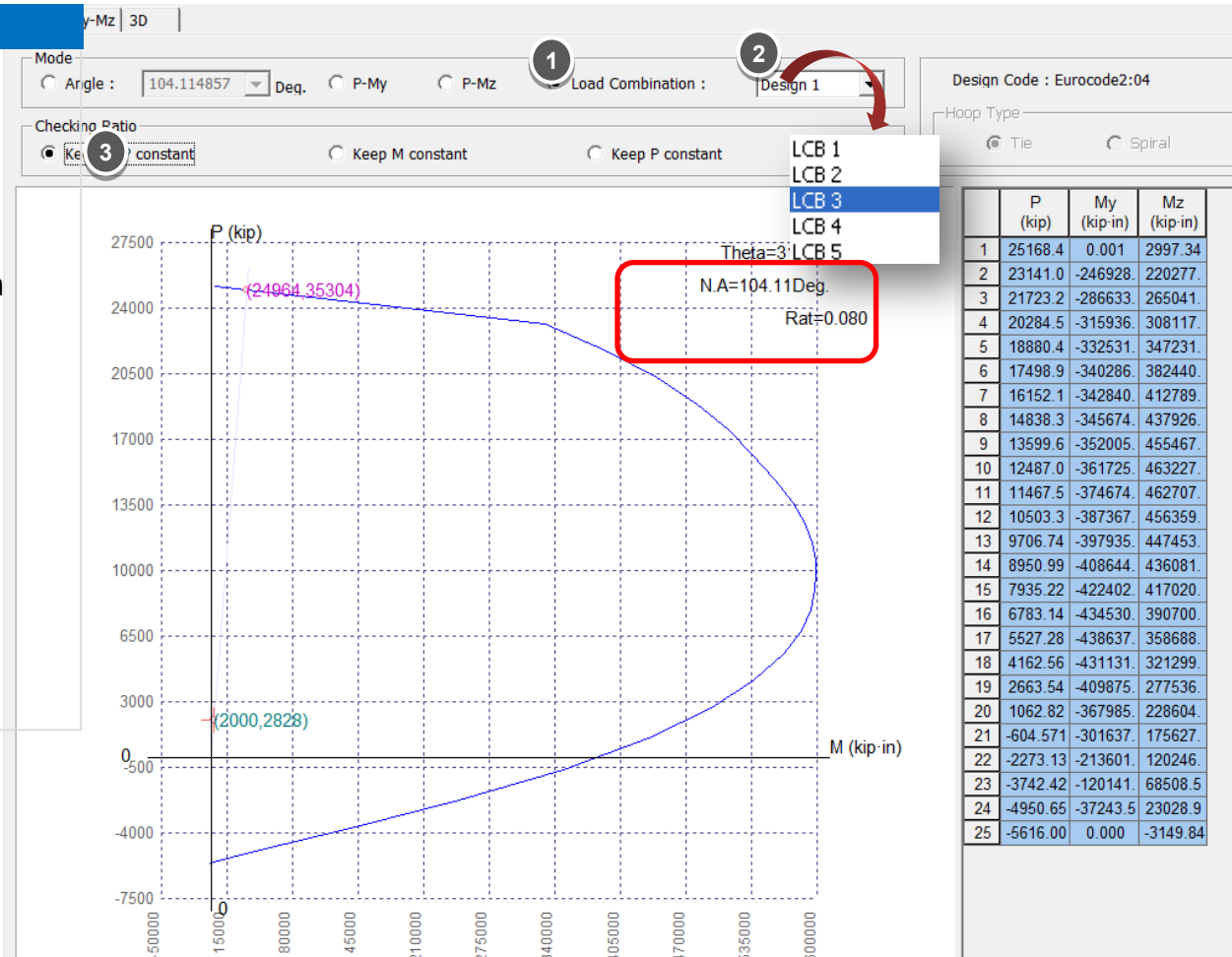
② Select Design 1

Checking Ratio can be calculated in

③ three ways:

1. Keep M/P constant
2. Keep M constant
3. Keep P constant

Select Keep M/P constant

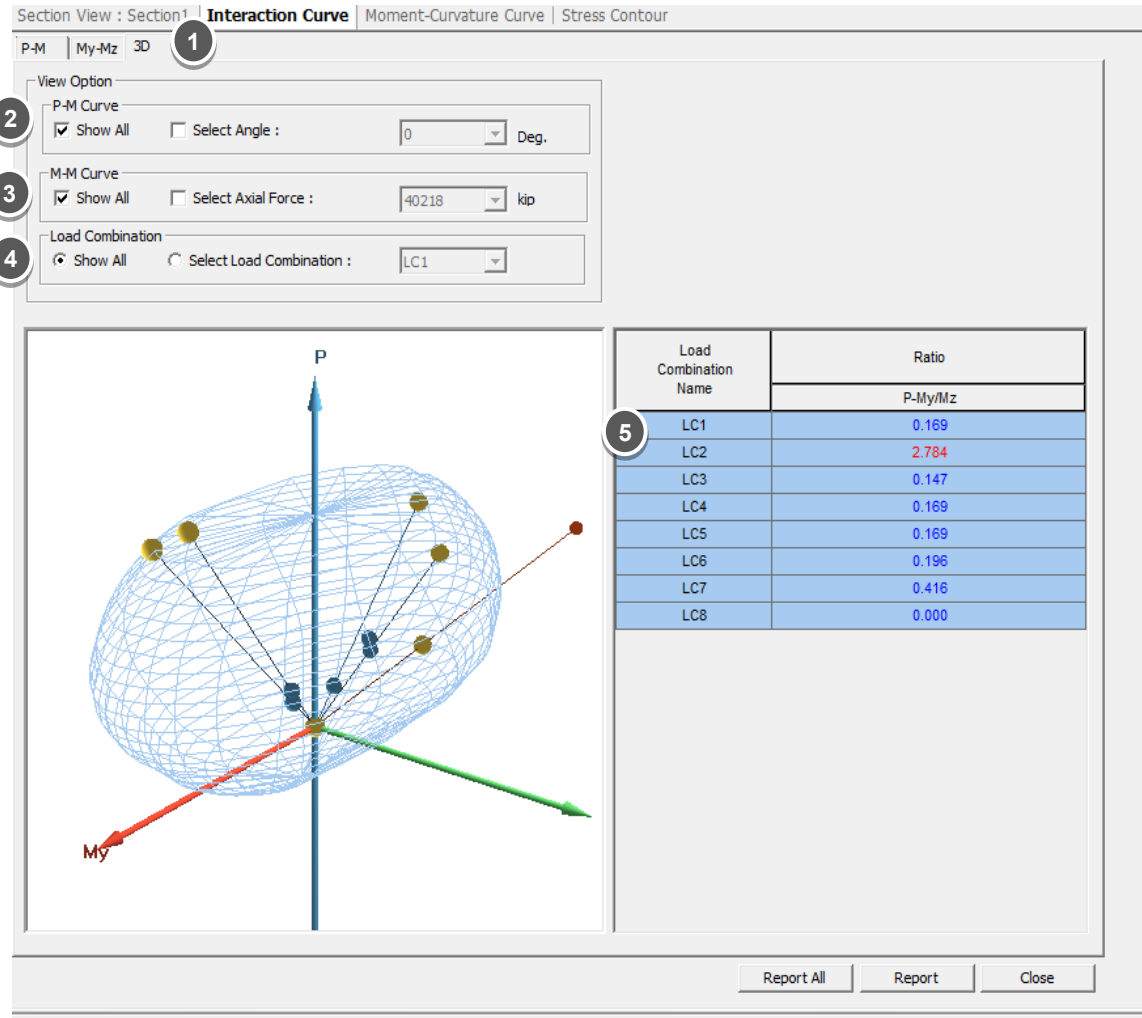


Results

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
- ❹ Select Show All in Load Combination
- ❺ The 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

① Click on “**Moment-Curvature**” tab.

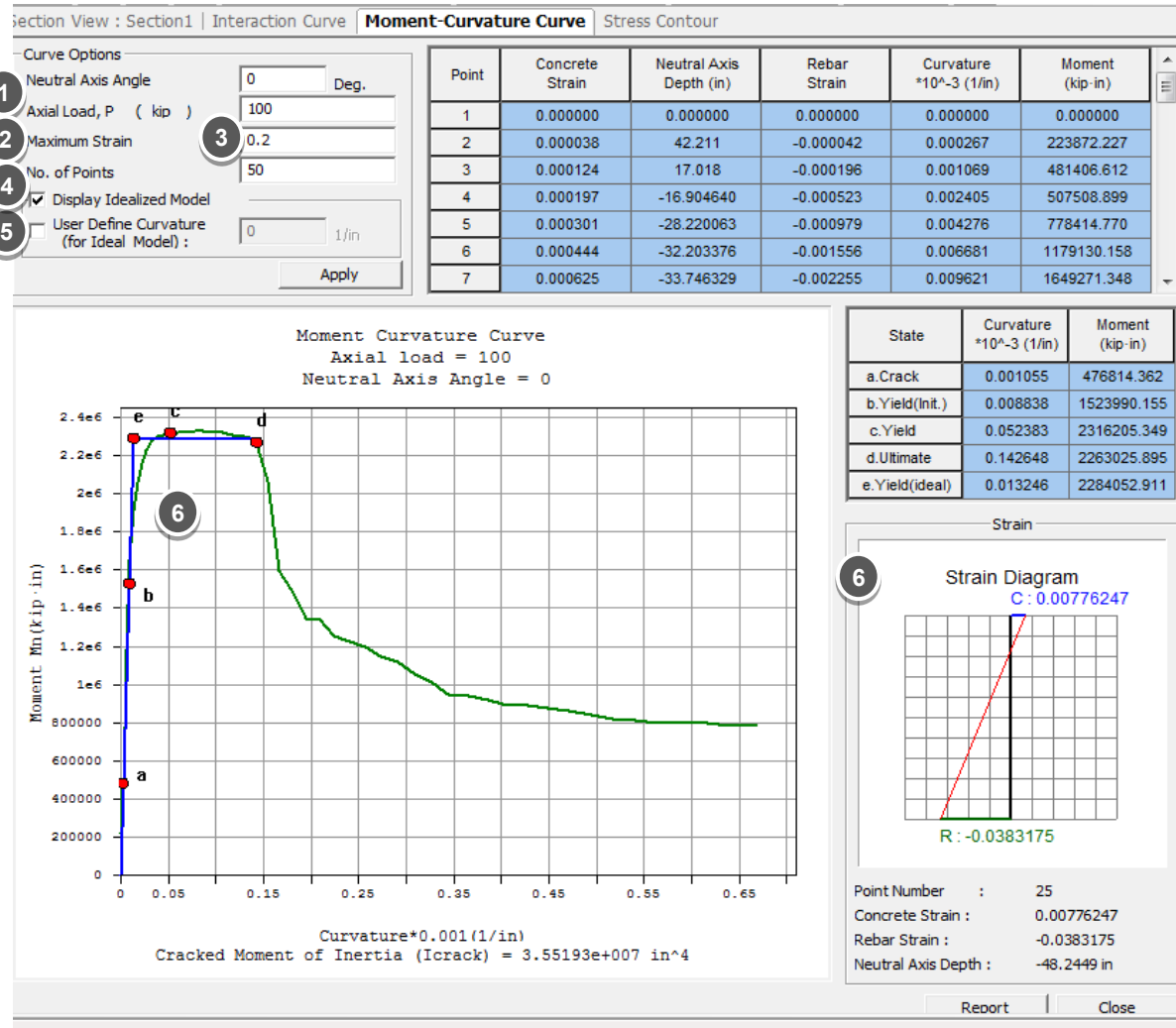
② Enter 100 kips as Axial Load P

③ Enter 0.2 as the Maximum Strain

④ Number of Points: 50

⑤ Display Idealized Model → Apply

⑥ 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.

