

Pier Modeling Example

- *Bridging Your Innovations to Realities*

Training material



midas **Civil**



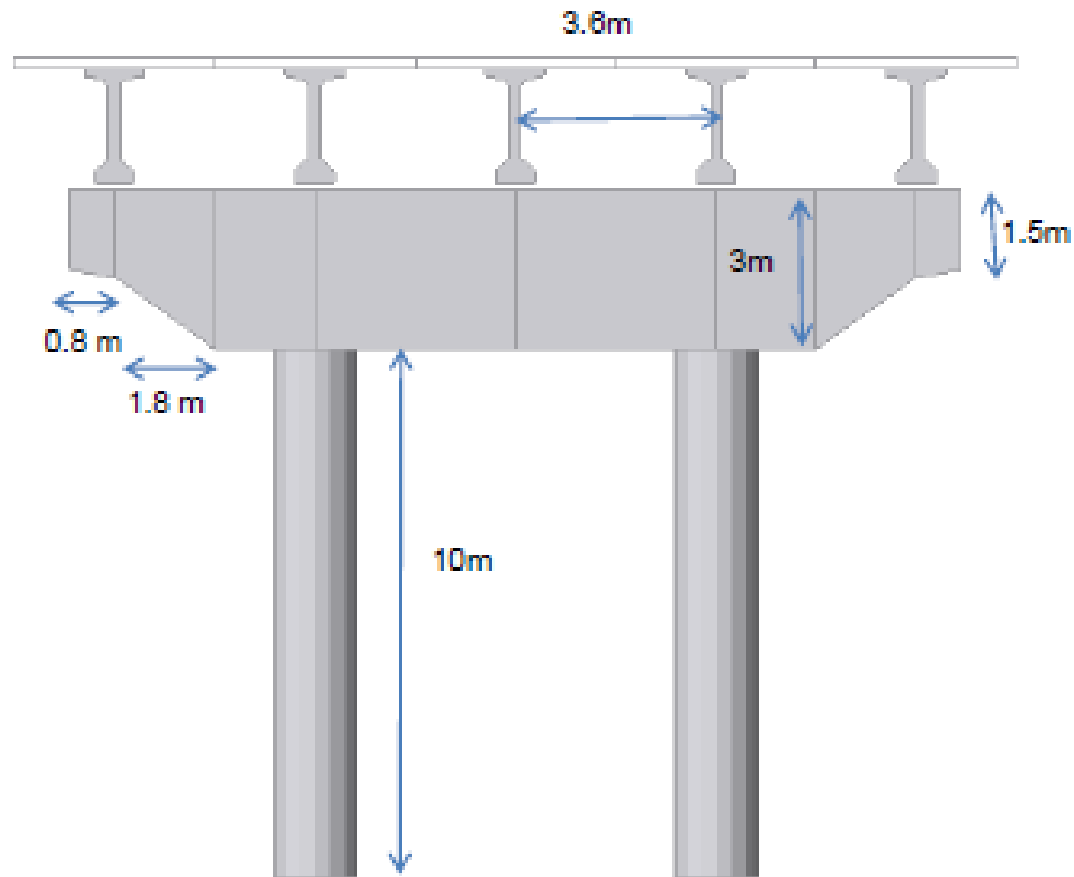
Overview: Pier Modeling

1. Response Spectrum Analysis

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



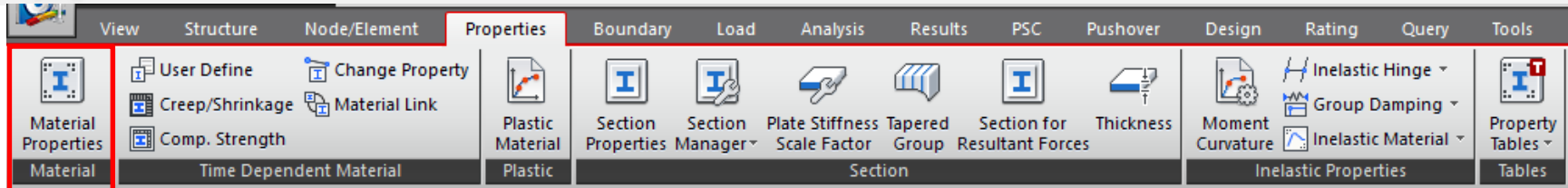
1. Introduction: Pier Modeling



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

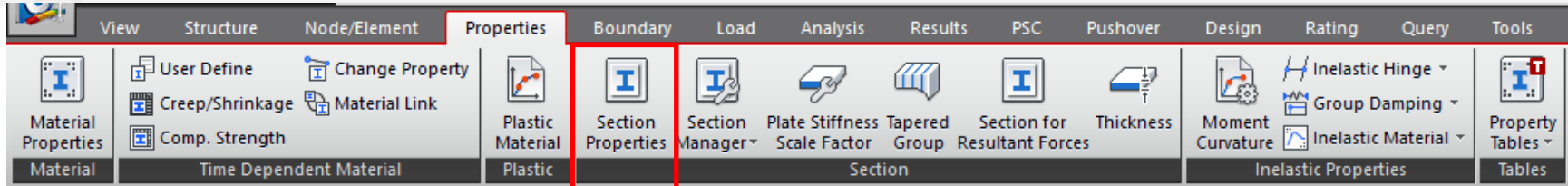
Properties > Material Properties

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:

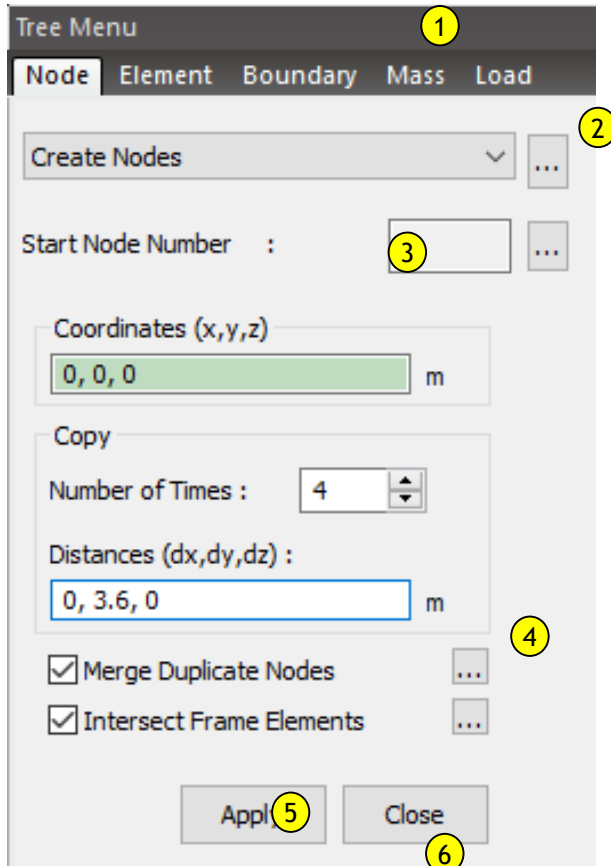
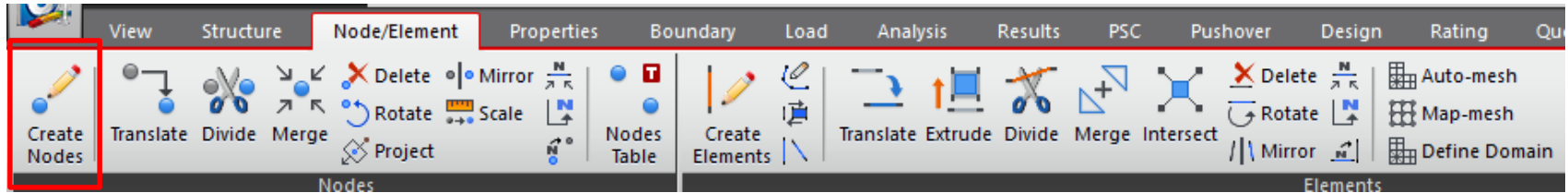
1. Select DB/USER tab
2. Select Solid Rectangle
3. Enter 3 m X 1.5 m
4. Set Offset at Center Top
5. Click Apply

Pier:

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



4. Modeling



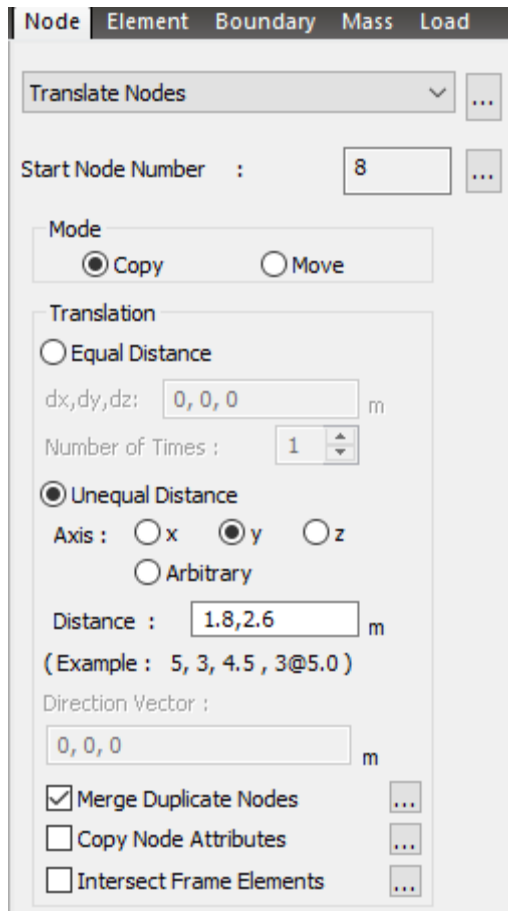
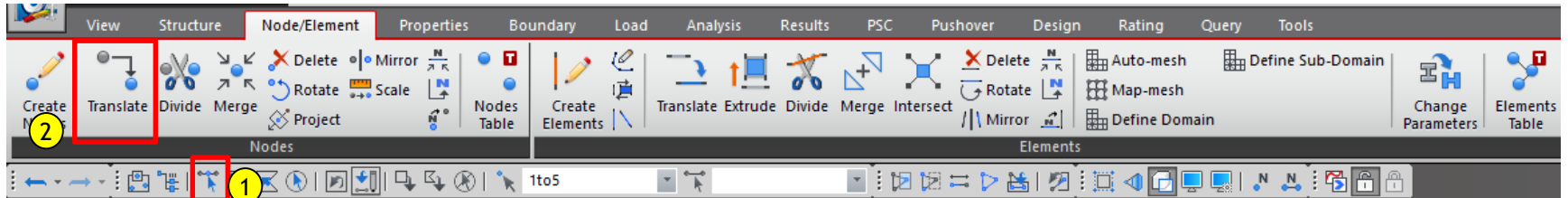
Node/Element > Create Nodes

Add Section

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

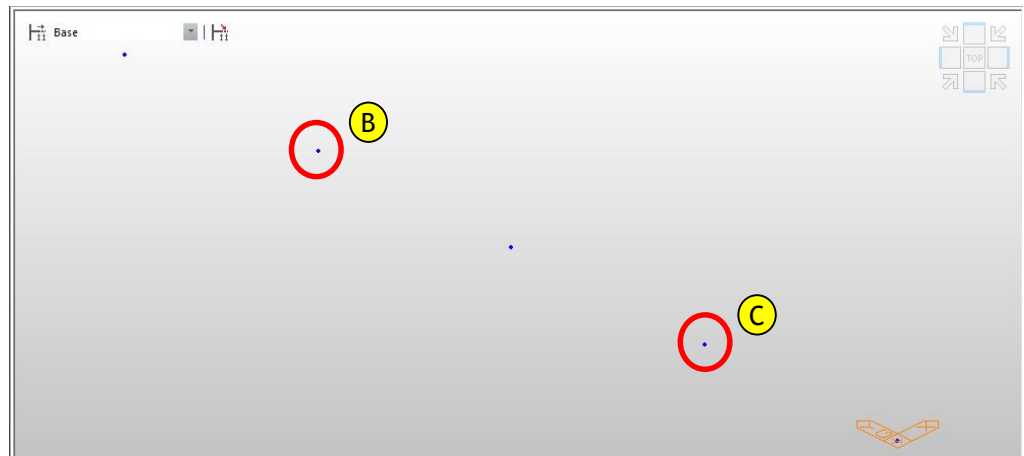


4. Modeling



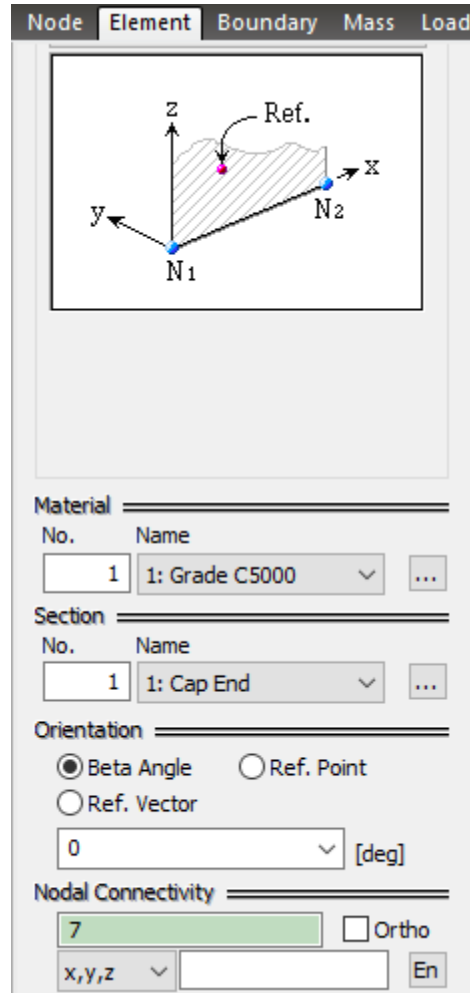
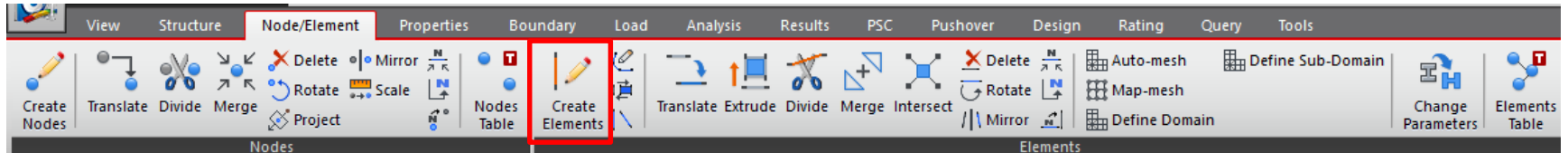
Node/Element > Translate Nodes

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



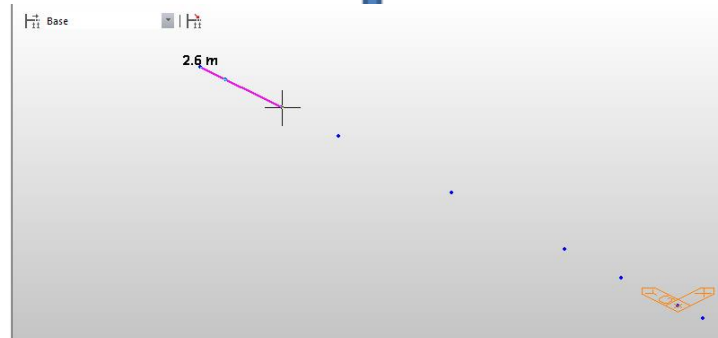
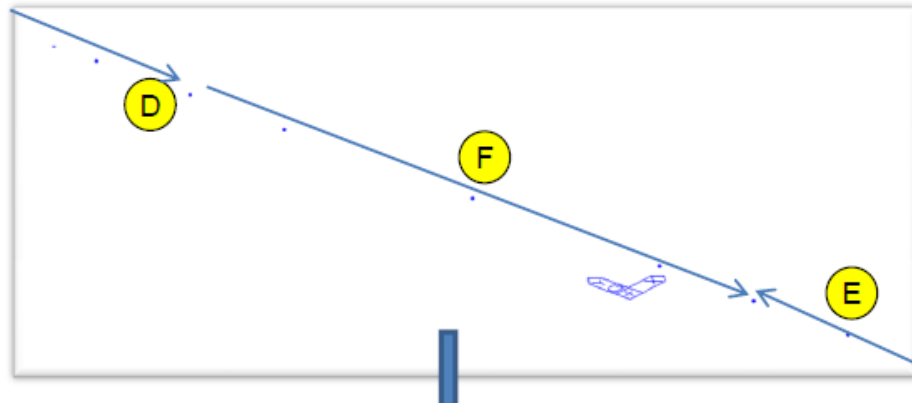


4. Modeling



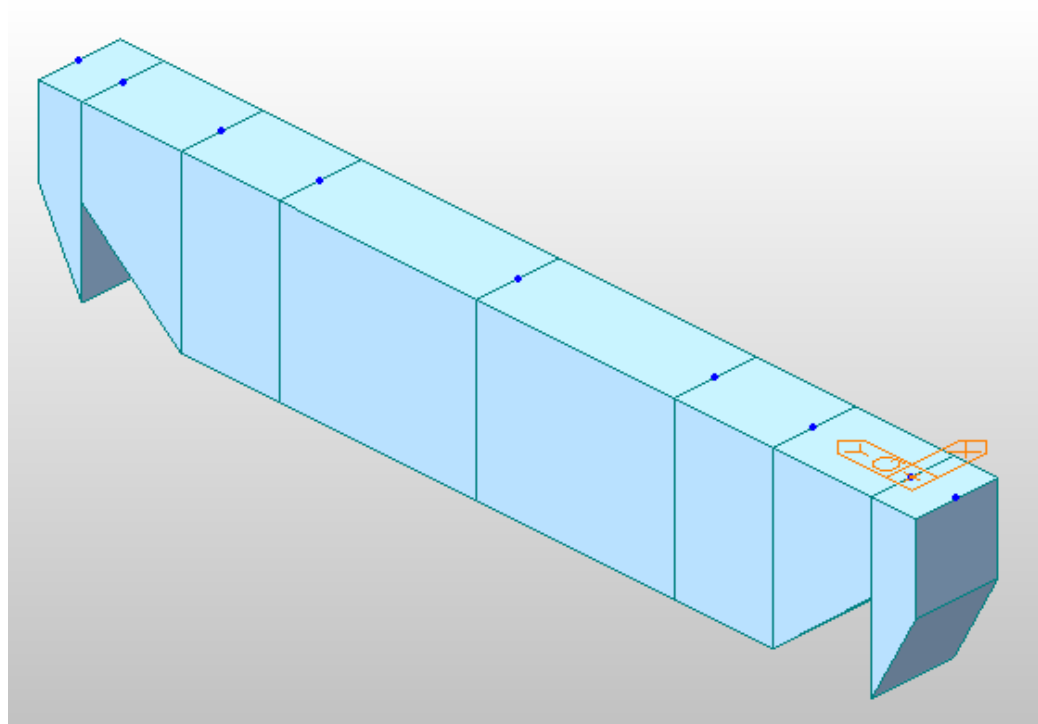
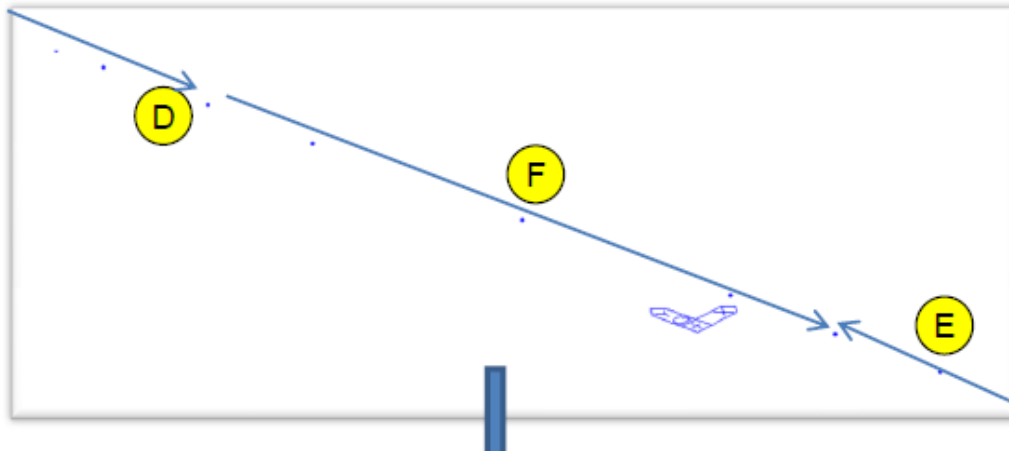
Node/Element > Create Elements

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]



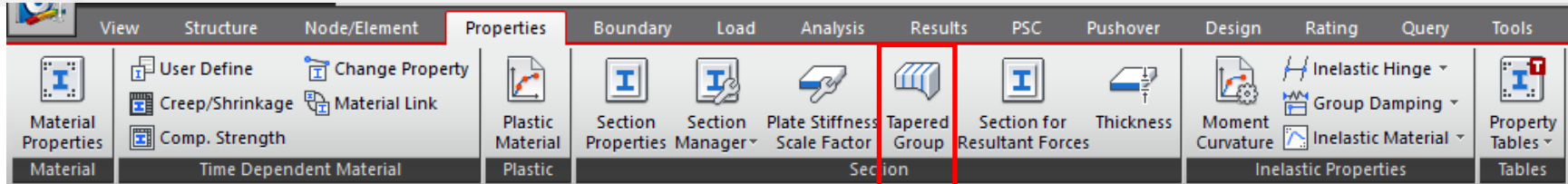


4. Modeling



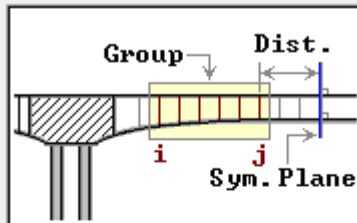


4. Modeling



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

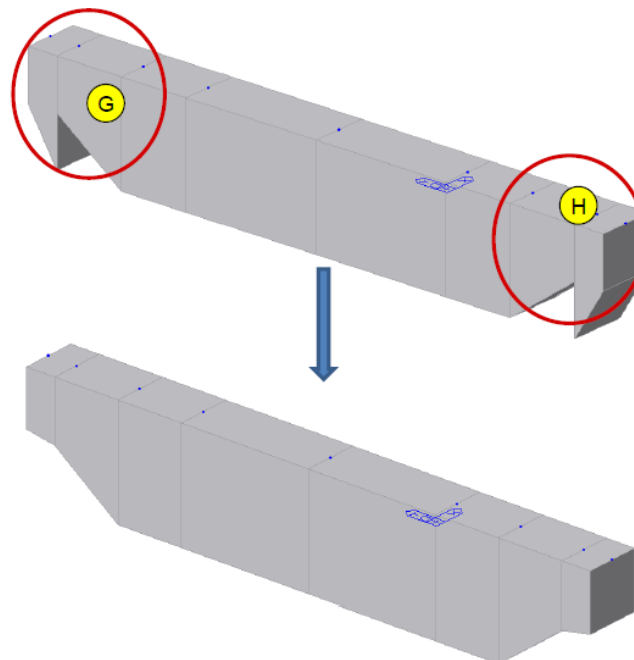
Symmetric Plane

From : ☐ i ☐ j

Distance : 0 m

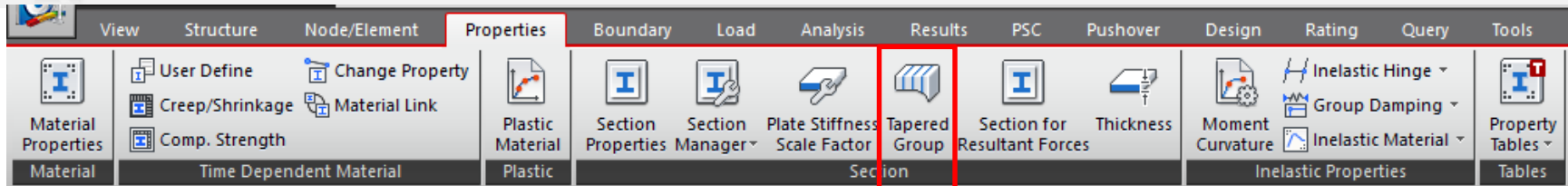
Properties > Tapered Group

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)



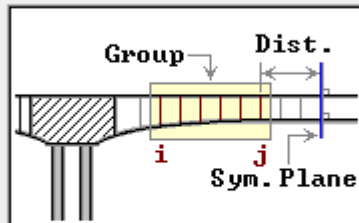


4. Modeling



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

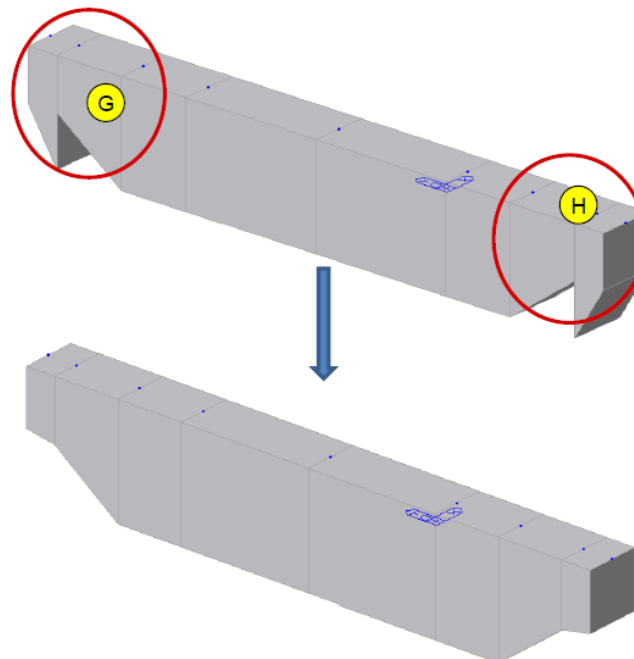
Symmetric Plane

From : ☐ i ☐ j

Distance : 0 m

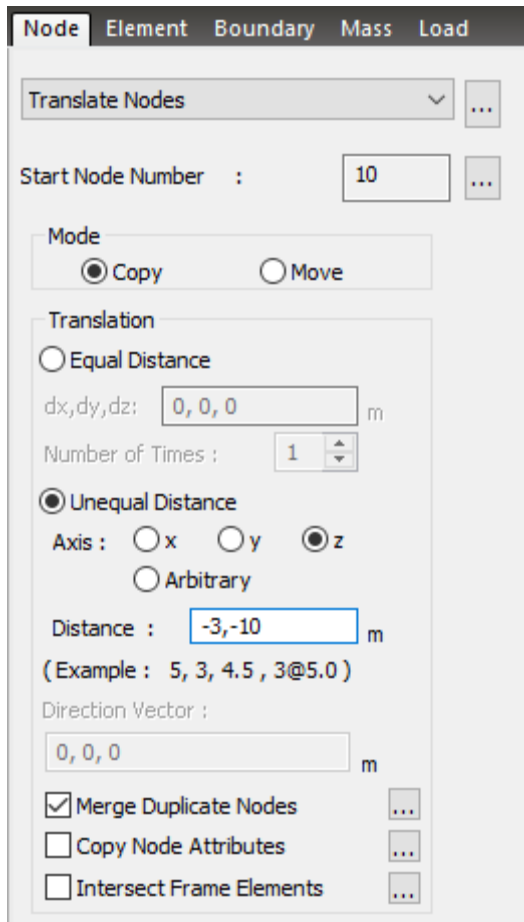
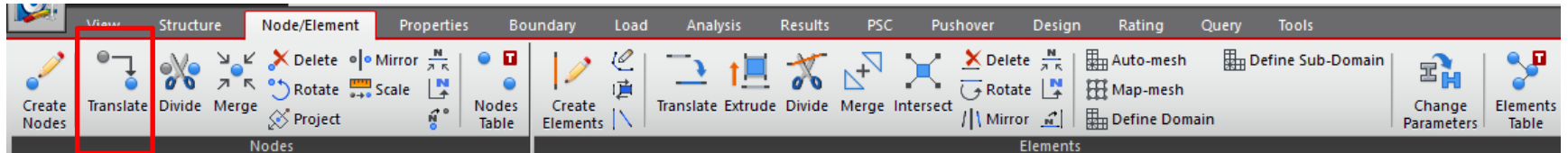
Properties > Tapered Group

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)



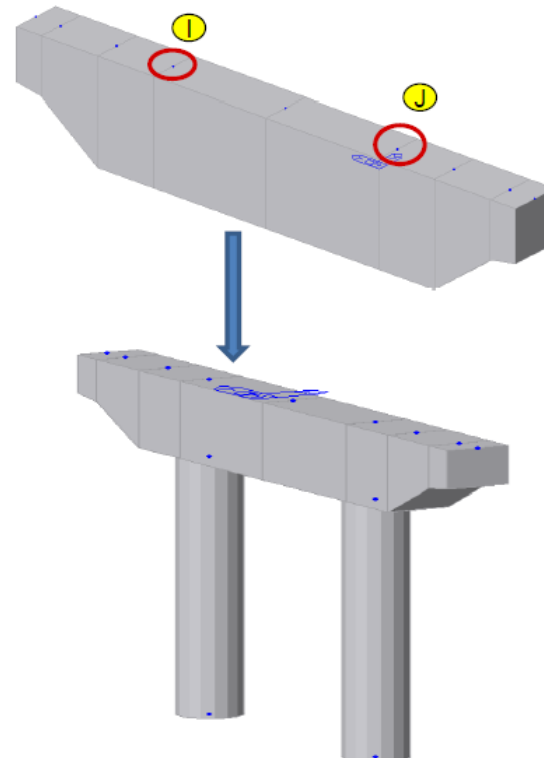


4. Modeling



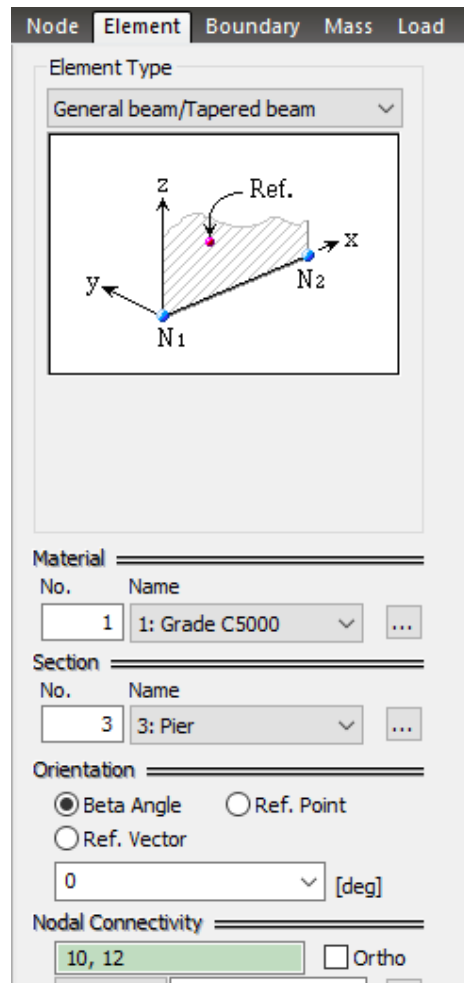
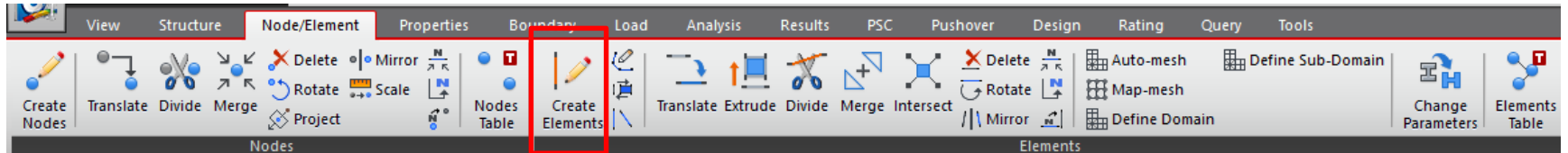
Node/Element>Translate

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)



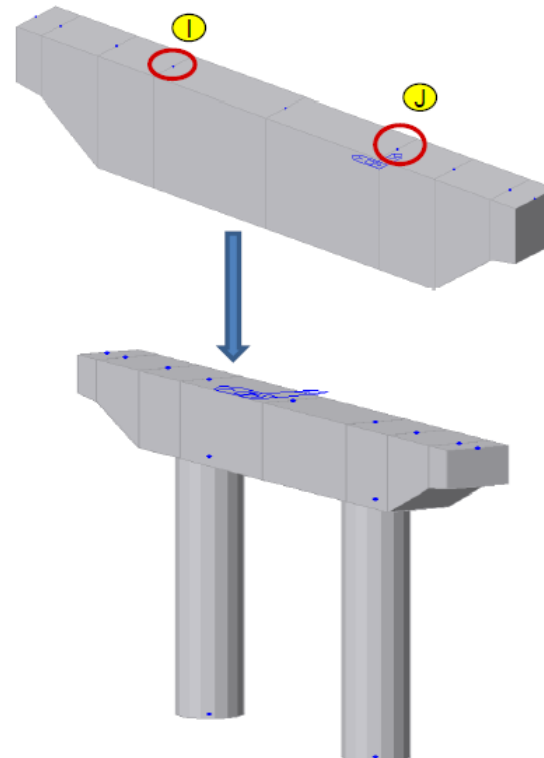


4. Modeling



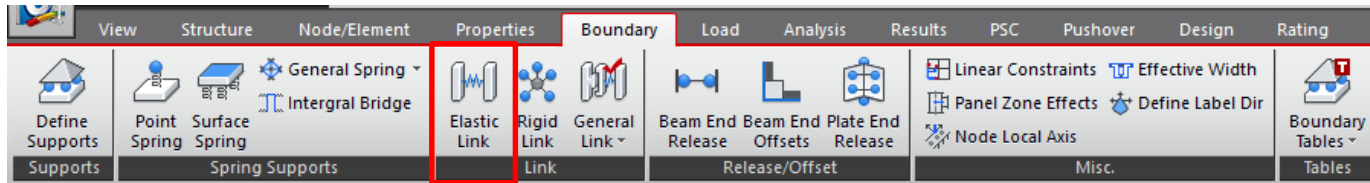
Node/Element>Translate

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



Node Element Boundary Mass Load

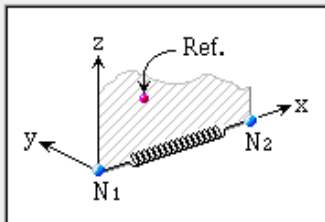
Options

☒ Add

☐ Delete

Elastic Link Data

Type:



SDx: tonf/m

SDy: tonf/m

SDz: tonf/m

SRx: tonf*m/[rad]

SRy: tonf*m/[rad]

SRz: tonf*m/[rad]

☐ Shear Spring Location

Distance Ratio From End I

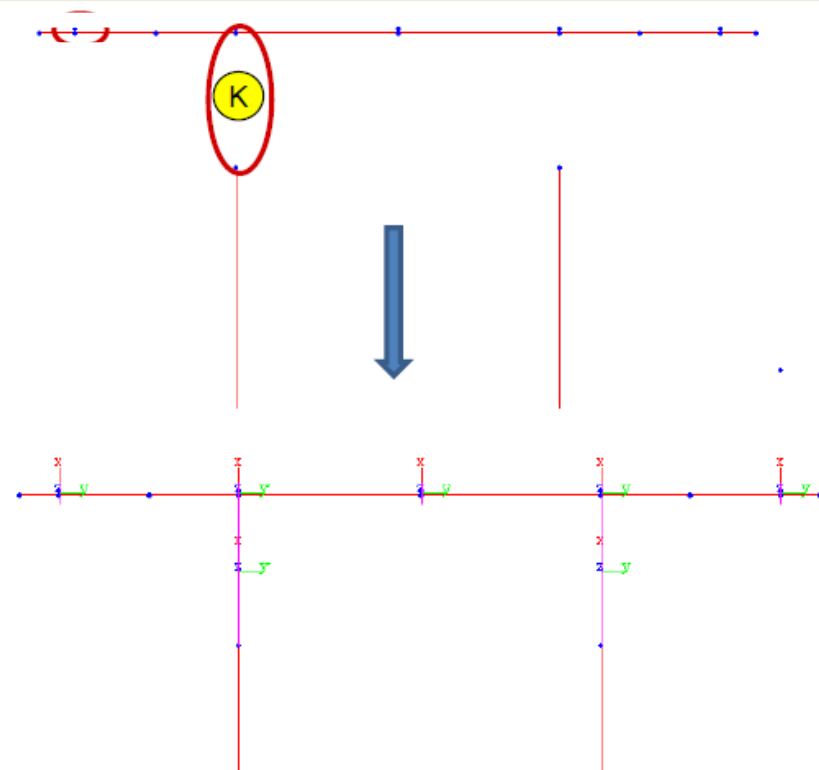
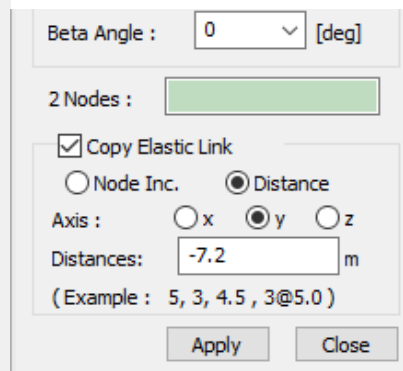
SDy: SDz:

Beta Angle: [deg]

2 Nodes:

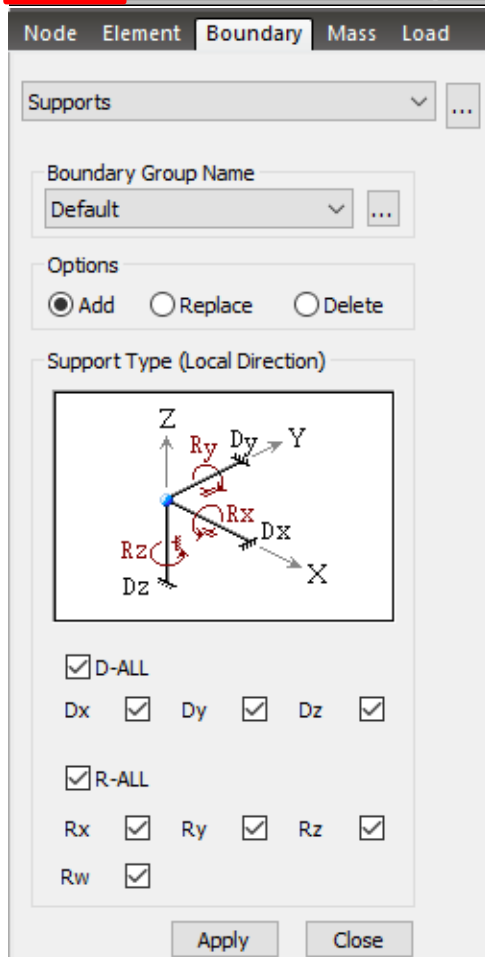
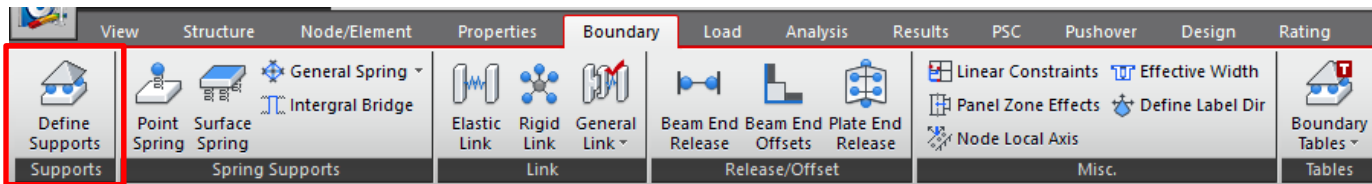
Boundary > Elastic Link

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



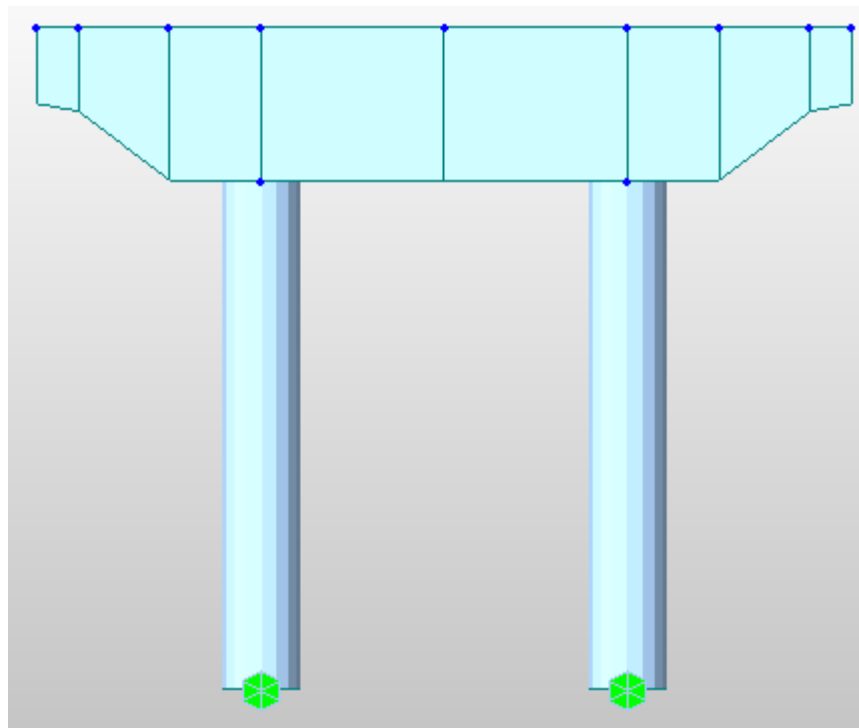


4. Modeling



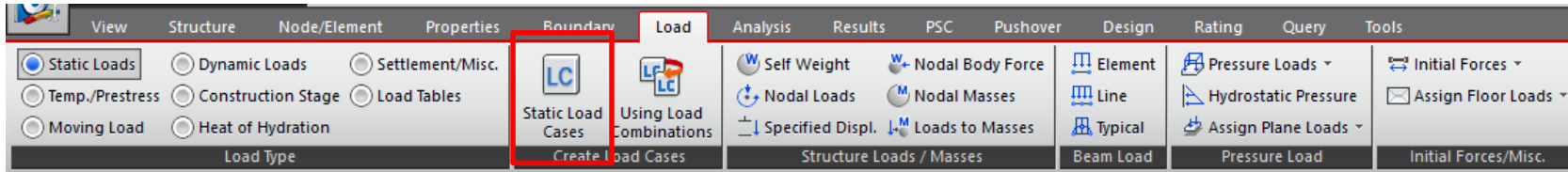
Boundary > Define Supports

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.



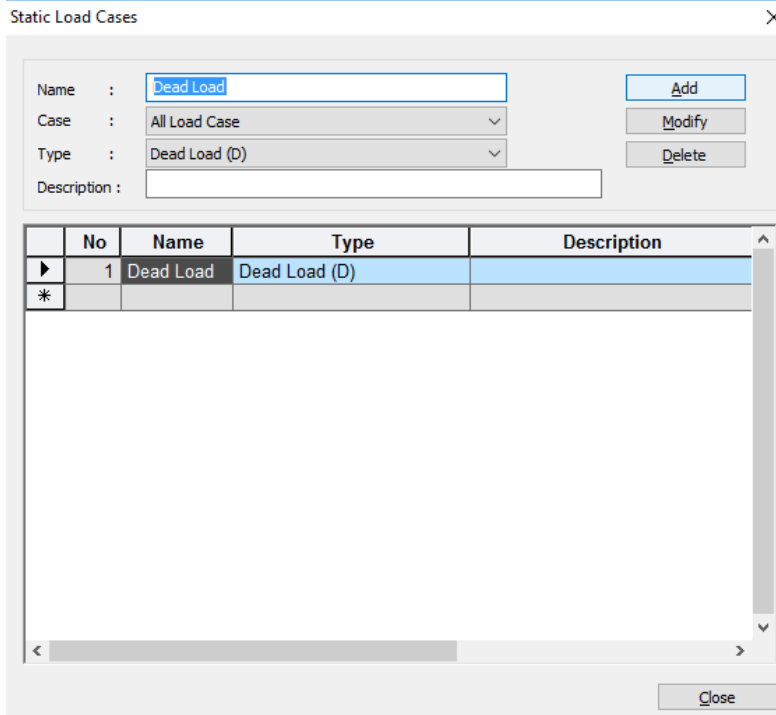


4. Modeling



Load > Static Load > Static Load Cases

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





4. Modeling

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

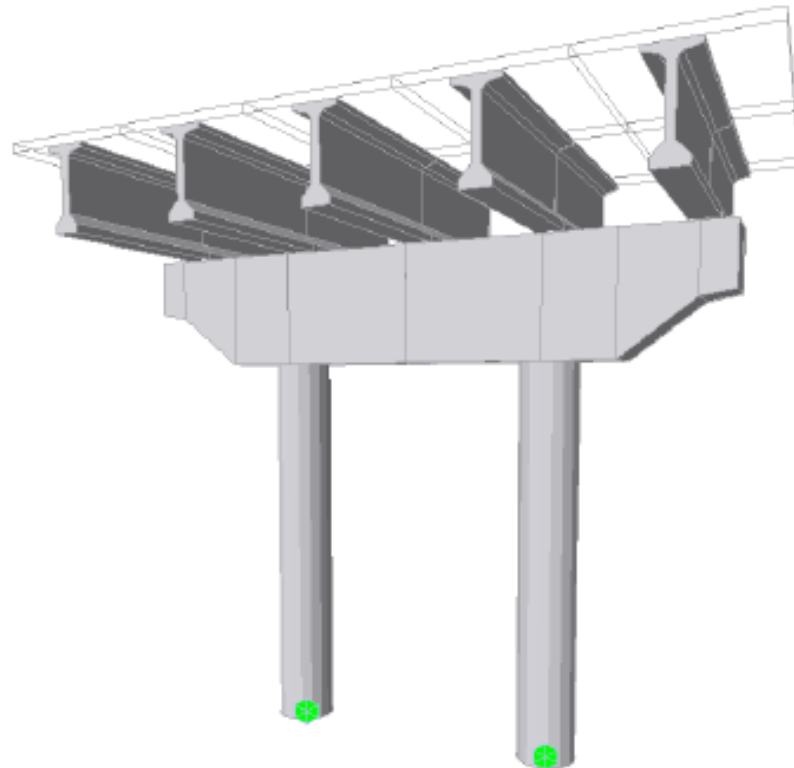
Load > Static Loads > Nodal Loads

- 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/>