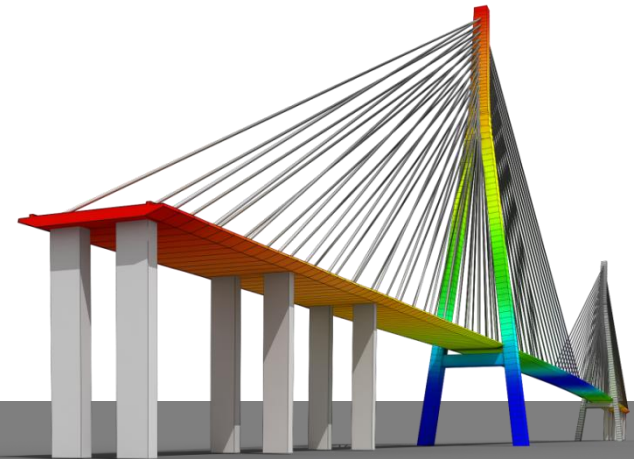


# Prestressing Plate Elements

## Midas Civil

- *Bridging Your Innovations to Realities*



## ► Introduction

### **Prestressing Plate Elements in Midas Civil.**

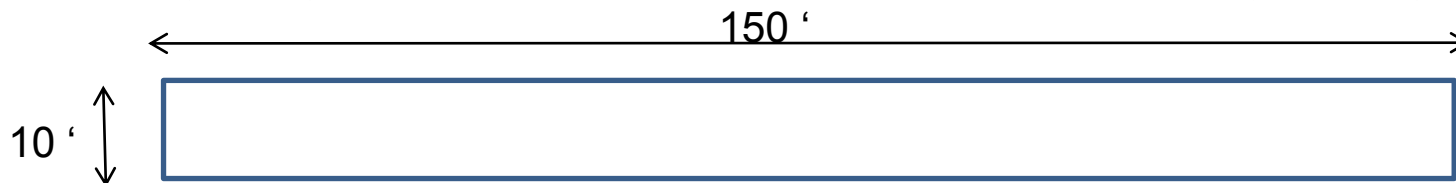
In Midas Civil the prestressing cannot be applied to the plate elements and hence the sole purpose of this tutorial is model the prestressing in the plate elements through a workaround.

For exhibiting this work around, we will model the prestressing in the plate elements in the transverse and the longitudinal direction.

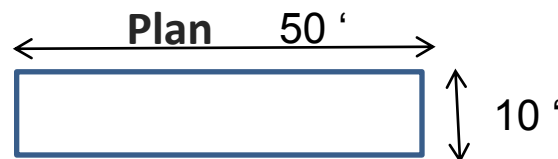
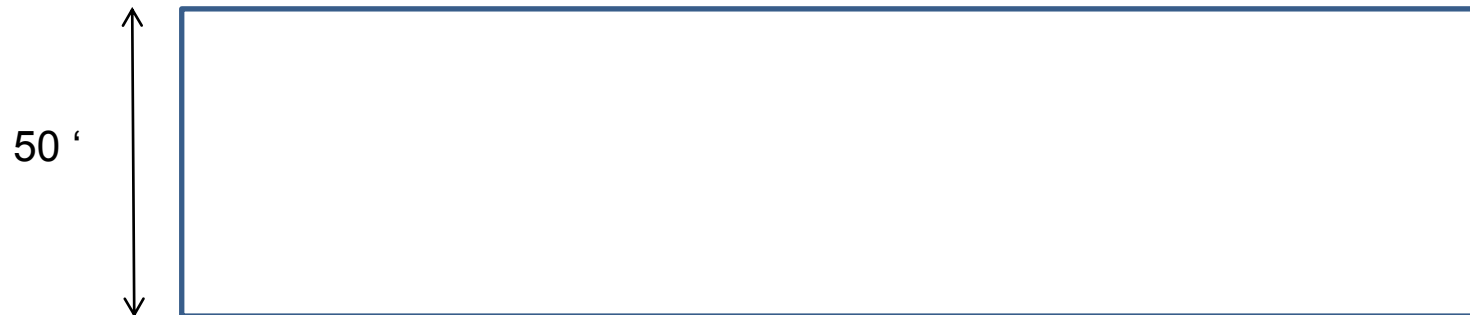
## ► Introduction

### Bridge Introduction

The following are the drawings of the bridge. For this tutorial the drawings are kept simple so that the user can understand the modeling technique.



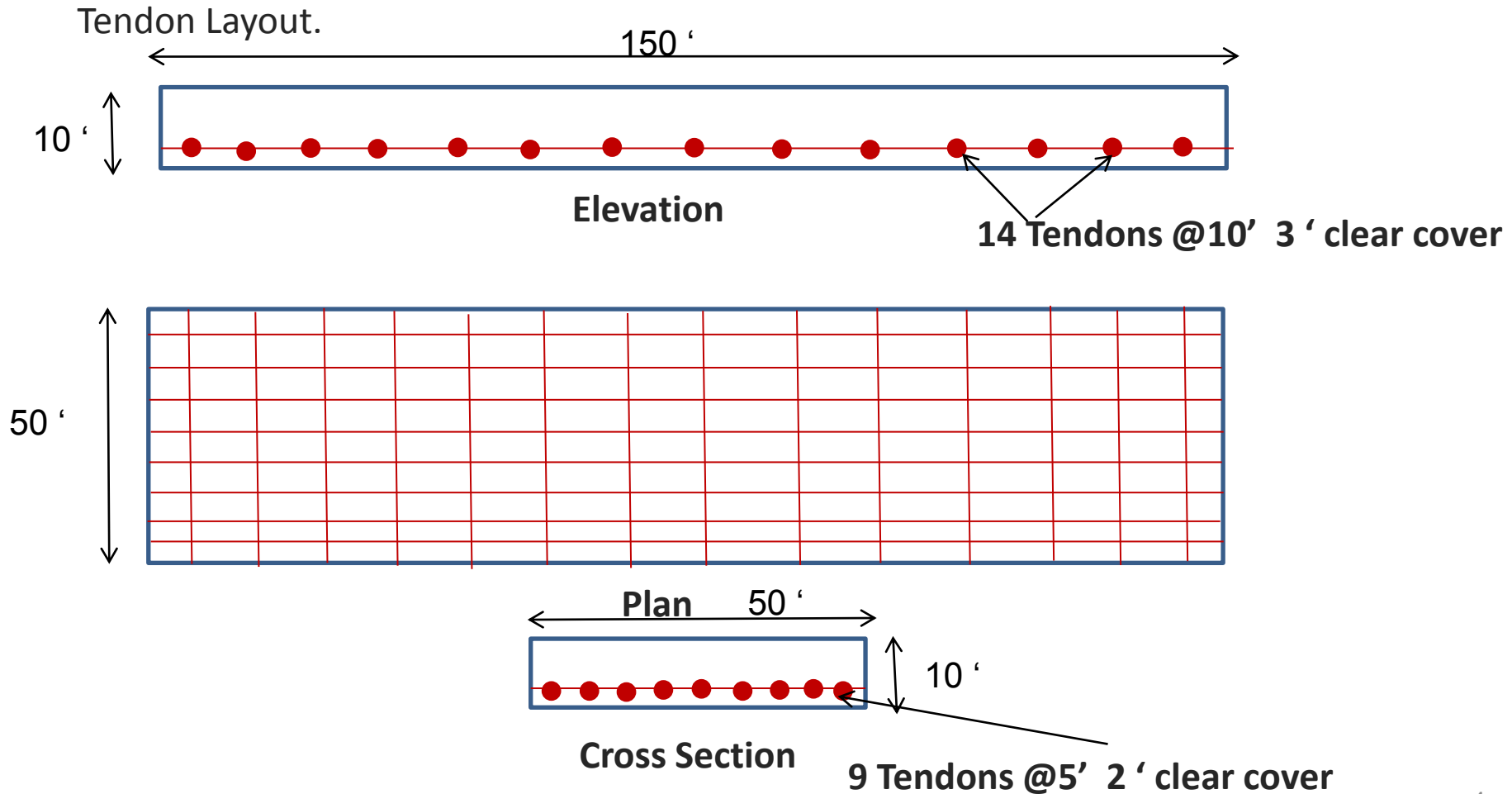
**Elevation**



**Cross Section**

## ► Introduction

### Bridge Introduction



## ► Introduction

### Modeling:

#### Tendon Modeling:

For modeling the tendons we should keep the following in mind:

1. We should have beams at the location of the tendons.
2. The stiffness of the beams must be  $1/1000$  of the surrounding elements.
3. The tendons can be only post tensioned type and never pre tensioned. Pre tensioned tendons would lead to numerical instability in the program as the elastic shortening loss will be very high.

Hence the limitation of this method is that only the post tensioned effects of tendons can be modeled.

## ► Introduction

### Modeling:

1. Deck Modeling: We will model the bridge using the plate elements and we will provide the thickness as 10 feet.
1. Tendon Modeling: Since we cannot model the prestressing tendons in the plate elements, we will model the girders at the location of the tendons and then assign the tendons to the beams. The beams will be provided a nominal cross section and hence it would not interfere with the stiffness of the bridge as a whole. The prestressing will be transferred to the plate elements from the beams.

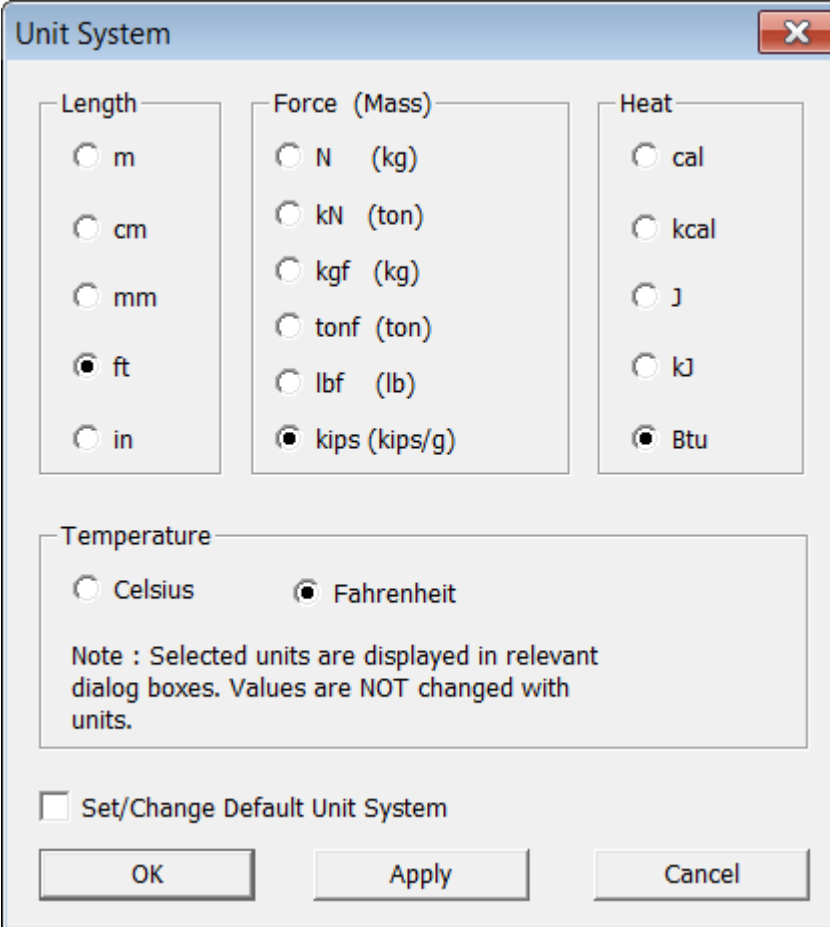
## ▶ 2. Setting Units

### Setting of Units:

**File → New Project**

**Tools → Unit System**

1. Length → ft
2. Force → kips
3. Click → OK



The image shows a 'Unit System' dialog box with three columns of radio buttons for selecting units. The 'Length' column has options for m, cm, mm, ft (selected), and in. The 'Force (Mass)' column has options for N (kg), kN (ton), kgf (kg), tonf (ton), lbf (lb), and kips (kips/g) (selected). The 'Heat' column has options for cal, kcal, J, kJ, and Btu (selected). Below these columns is a 'Temperature' section with radio buttons for Celsius and Fahrenheit (selected). A note states: 'Note : Selected units are displayed in relevant dialog boxes. Values are NOT changed with units.' At the bottom, there is a checkbox for 'Set/Change Default Unit System' and three buttons: 'OK', 'Apply', and 'Cancel'.

Length	Force (Mass)	Heat
<input type="radio"/> m	<input type="radio"/> N (kg)	<input type="radio"/> cal
<input type="radio"/> cm	<input type="radio"/> kN (ton)	<input type="radio"/> kcal
<input type="radio"/> mm	<input type="radio"/> kgf (kg)	<input type="radio"/> J
<input checked="" type="radio"/> ft	<input type="radio"/> tonf (ton)	<input type="radio"/> kJ
<input type="radio"/> in	<input type="radio"/> lbf (lb)	<input checked="" type="radio"/> Btu
	<input checked="" type="radio"/> kips (kips/g)	

Temperature

☐ Celsius ☒ Fahrenheit

Note : Selected units are displayed in relevant dialog boxes. Values are NOT changed with units.

☐ Set/Change Default Unit System

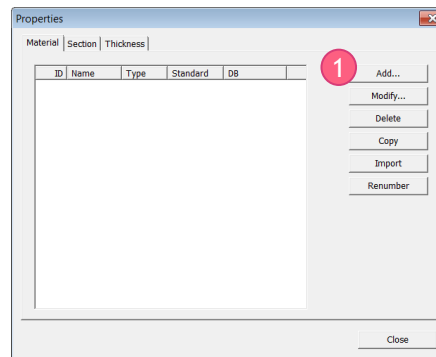
OK Apply Cancel

# ▶ 3. Material Definition

## Definition of Material :

### Model → Properties → Material

1. Click Add
2. Type of Design → Concrete
3. Name → Grade C4000
4. Standard → ASTM ( RC)
5. DB → Grade C4000
6. Click → OK



The screenshot shows the 'Material Data' dialog box. Red circles with numbers 2 through 6 highlight specific fields: 2 points to the 'Type of Design' dropdown (set to 'Concrete'); 3 points to the 'Name' field (set to 'Grade C4000'); 4 points to the 'Concrete Standard' dropdown (set to 'ASTM(RC)'); 5 points to the 'Concrete DB' dropdown (set to 'Grade C4000'); and 6 points to the 'OK' button at the bottom right.

**General**  
Material ID: 1  
Name: Grade C4000

**Elasticity Data**  
Type of Design: Concrete  
Steel Standard:   
DB:   
Concrete Standard: ASTM(RC)  
Code:   
DB: Grade C4000

**Type of Material**  
☒ Isotropic ☐ Orthotropic

**Steel**  
Modulus of Elasticity : 0.0000e+000 kips/in<sup>2</sup>  
Poisson's Ratio : 0  
Thermal Coefficient : 0.0000e+000 1/[F]  
Weight Density : 0 kips/in<sup>3</sup>  
☐ Use Mass Density: 0 kips/in<sup>3</sup>/q

**Concrete**  
Modulus of Elasticity : 3.6441e+003 kips/in<sup>2</sup>  
Poisson's Ratio : 0.2  
Thermal Coefficient : 5.0000e-006 1/[F]  
Weight Density : 8.681e-005 kips/in<sup>3</sup>  
☐ Use Mass Density: 2.248e-007 kips/in<sup>3</sup>/q

**Plasticity Data**  
Plastic Material Name: NONE

**Thermal Transfer**  
Specific Heat : 0 Btu/kips·[F]  
Heat Conduction : 0 Btu/in·hr·[F]

Damping Ratio : 0.05

OK Cancel Apply

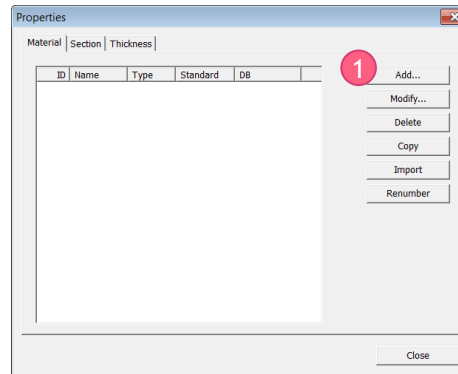


## ▶ 3. Material Definition

### Definition of Material :

#### Model → Properties → Material

1. Click Add
2. Type of Design → Steel
3. Name → A586-A1(Dia-L)
4. Standard → ASTM(S)
5. DB → A586-A1(Dia-L)
6. Click → OK



Material Data

General  
Material ID: 2 Name: A586-A1(Dia-L)

Elasticity Data  
2 of Design: Steel 4 Steel Standard: ASTM(S) 5 DB: A586-A1(Dia-L)  
Concrete Standard: Code: DB: [No Title]

Type of Material  
☒ Isotropic ☐ Orthotropic

Steel  
Modulus of Elasticity: 3.3120e+006 kips/ft²  
Poisson's Ratio: 0.3  
Thermal Coefficient: 6.5000e-006 1/[F]  
Weight Density: 0.4908 kips/ft³  
☐ Use Mass Density: 0.01525 kips/ft³/q

Concrete  
Modulus of Elasticity: 0.0000e+000 kips/ft²  
Poisson's Ratio: 0  
Thermal Coefficient: 0.0000e+000 1/[F]  
Weight Density: 0 kips/ft³  
☐ Use Mass Density: 0 kips/ft³/q

Plasticity Data  
Plastic Material Name: NONE

Thermal Transfer  
Specific Heat: 0 Btu/kips·[F]  
Heat Conduction: 0 Btu/ft·hr·[F]  
Damping Ratio: 0.05 6

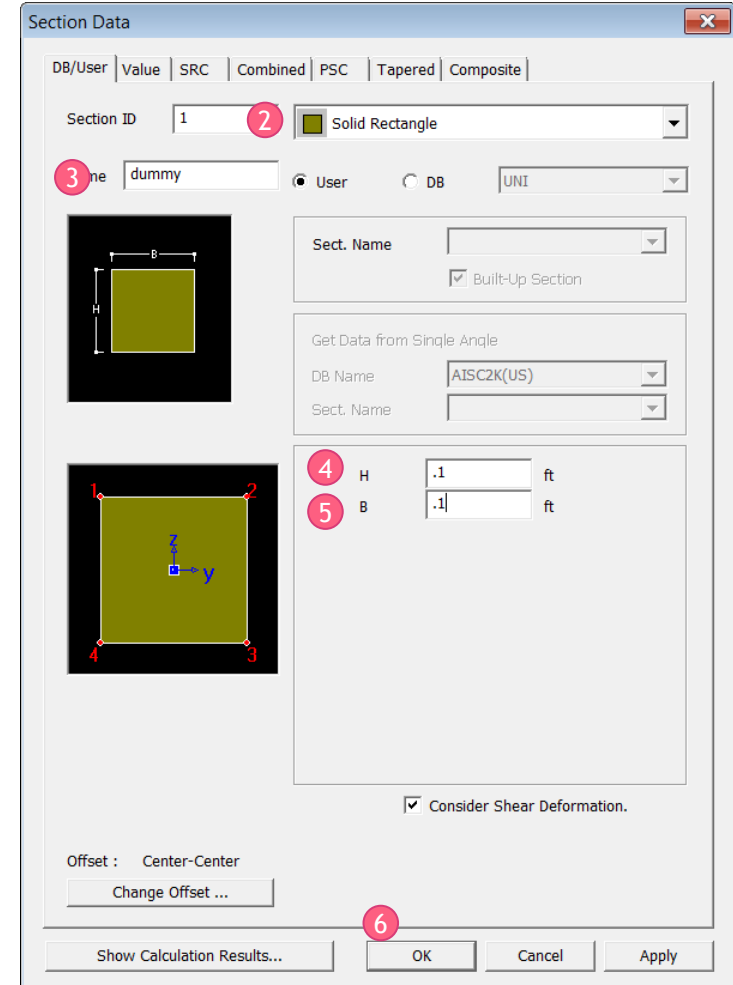
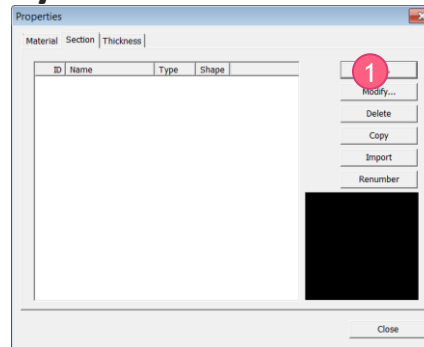
OK Cancel Apply

## ▶ 4. Section Definition

### Defining sections for dummy beams

#### Model → Properties → Section

1. Click Add
2. Type → Solid Rectangle
3. Name → dummy
4. H → .1
5. B → .1
6. Click → OK

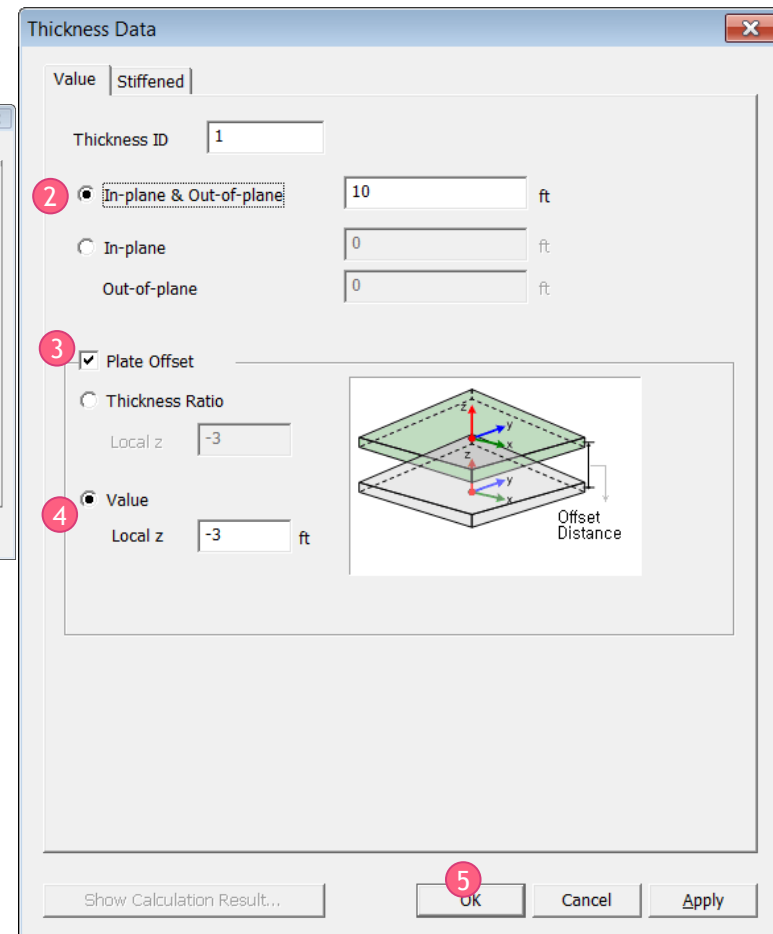
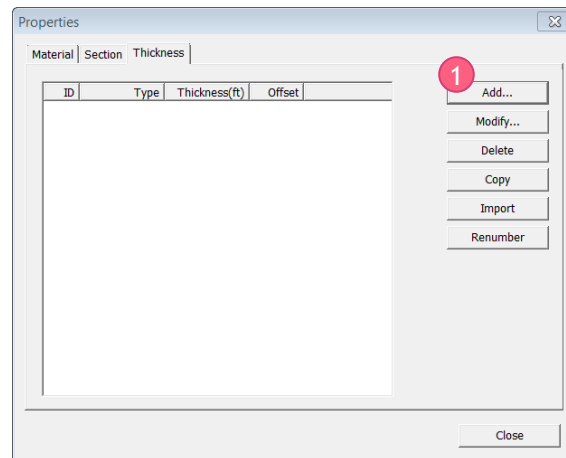


## ► 5. Thickness Definition

### Model → Properties →

#### Thickness

1. Click Add
2. In Plane and put of plane → 10
3. Plate offset → -3
4. Value → -3
5. Click → OK



## ▶ 6. Slab Modeling

### Model → Structure

#### wizard → Plate

1. B → 150
2. H → 50
3. Go to Edit Tab
4. Size of Divisions → 5
5. Go to Insert Tab
6. Alpha → 90
7. Click → OK

Plate Wizard

Input | Edit | Insert

Type1

B 150 ft

H 50 ft

R 0 ft

☐ Show Dimensions

Material 1 1: Grade C4000

Thickness 1 1: 10.0000

Redraw & Update Data

OK Close Apply

Plate Wizard

Input | Edit | Insert

Type2

Db 0 ft

Dh 0 ft

b 0 ft

h 0 ft

r 0 ft

☒ Size of Divisions

☐ Number of Divisions

Size 5 ft

m 1

n 1

☐ Show Sub-dimensions

Redraw & Update Data

OK Close Apply

Plate Wizard

Input | Edit | Insert

Insert Point

0, 0, 0

Rotations

Alpha 90

Beta 0

Gamma 0

☒ Merge Duplicate Nodes

☒ Intersect Frame Elements

Origin Point

☐ Show No. 1(0, 0, 0)

Redraw & Update Data

OK Close Apply

## ▶ 7. Dummy beam modeling

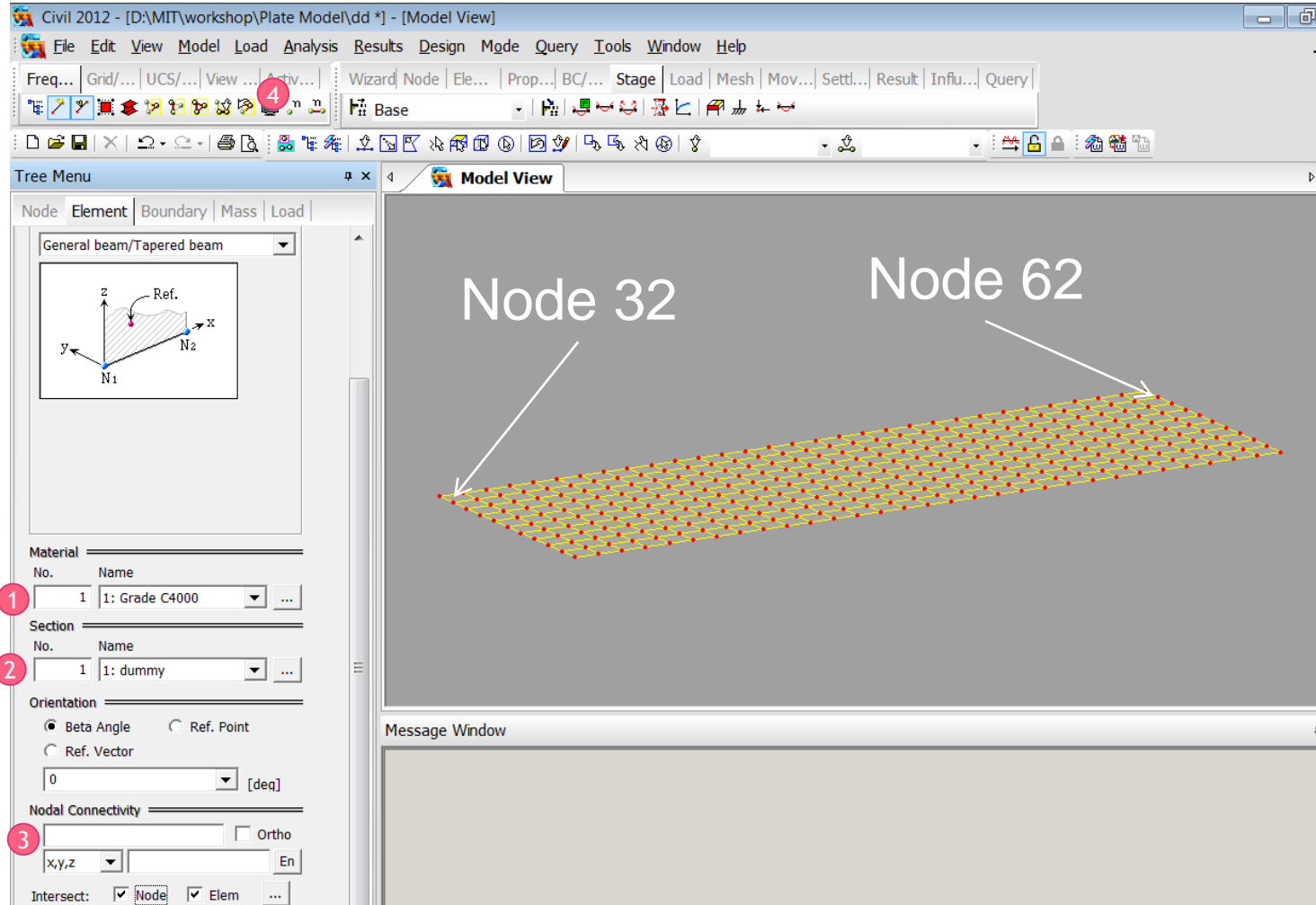
### Model → Elements →

#### Create elements

1. Material → 1: Grade c 4000
2. Section 1: dummy
3. Nodal connectivity

Click on it and it will turn green after it turns green Click on nodes 32 and 62


Note: the node number can be obtained from 4



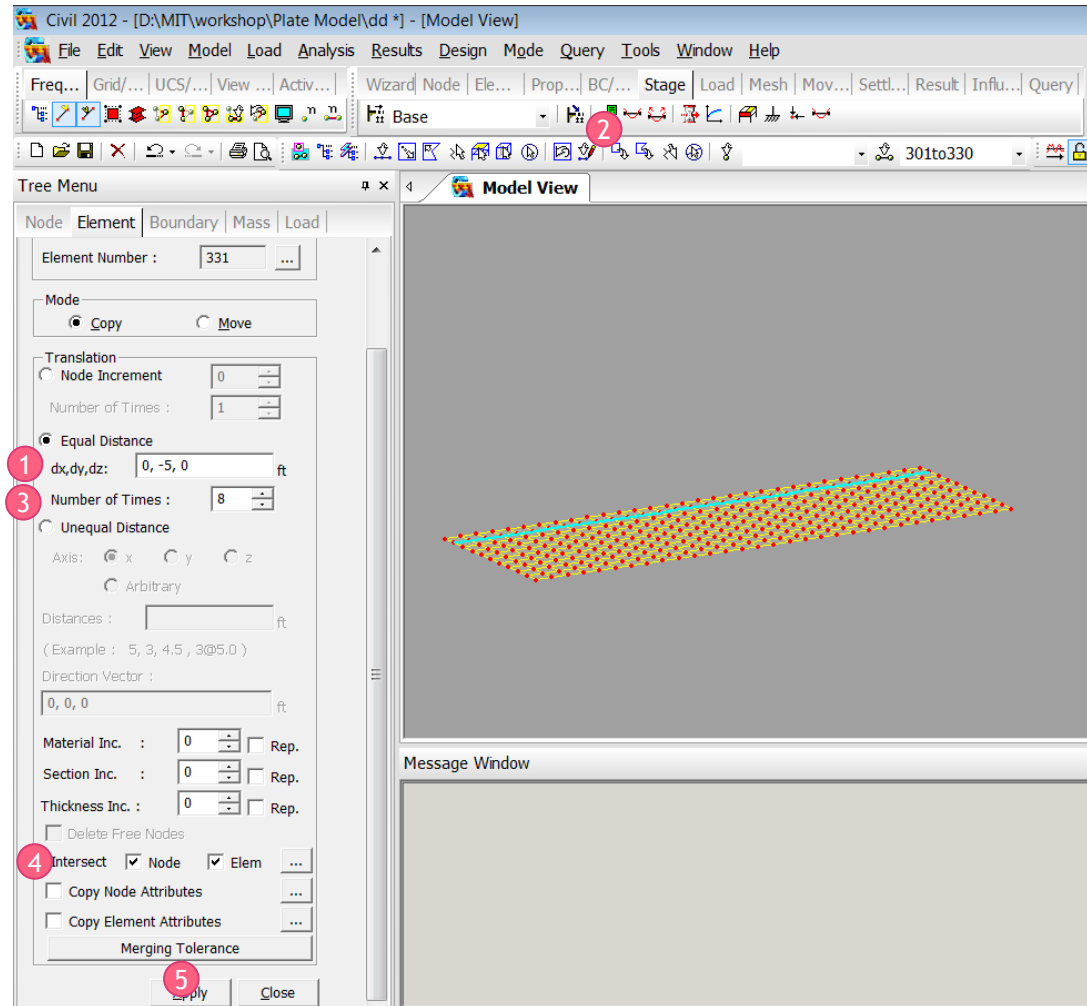
## ▶ 7. Dummy beam modeling

### Model → Elements →

#### Translate

1.  $Dx, dy, dz \rightarrow 0, -5, 0$
2. Click on  to select recently created beams
3. Number of Times  $\rightarrow 8$
4. Check on Intersect node and element
5. Click  $\rightarrow$  Apply

Note: the node number can be obtained from 4



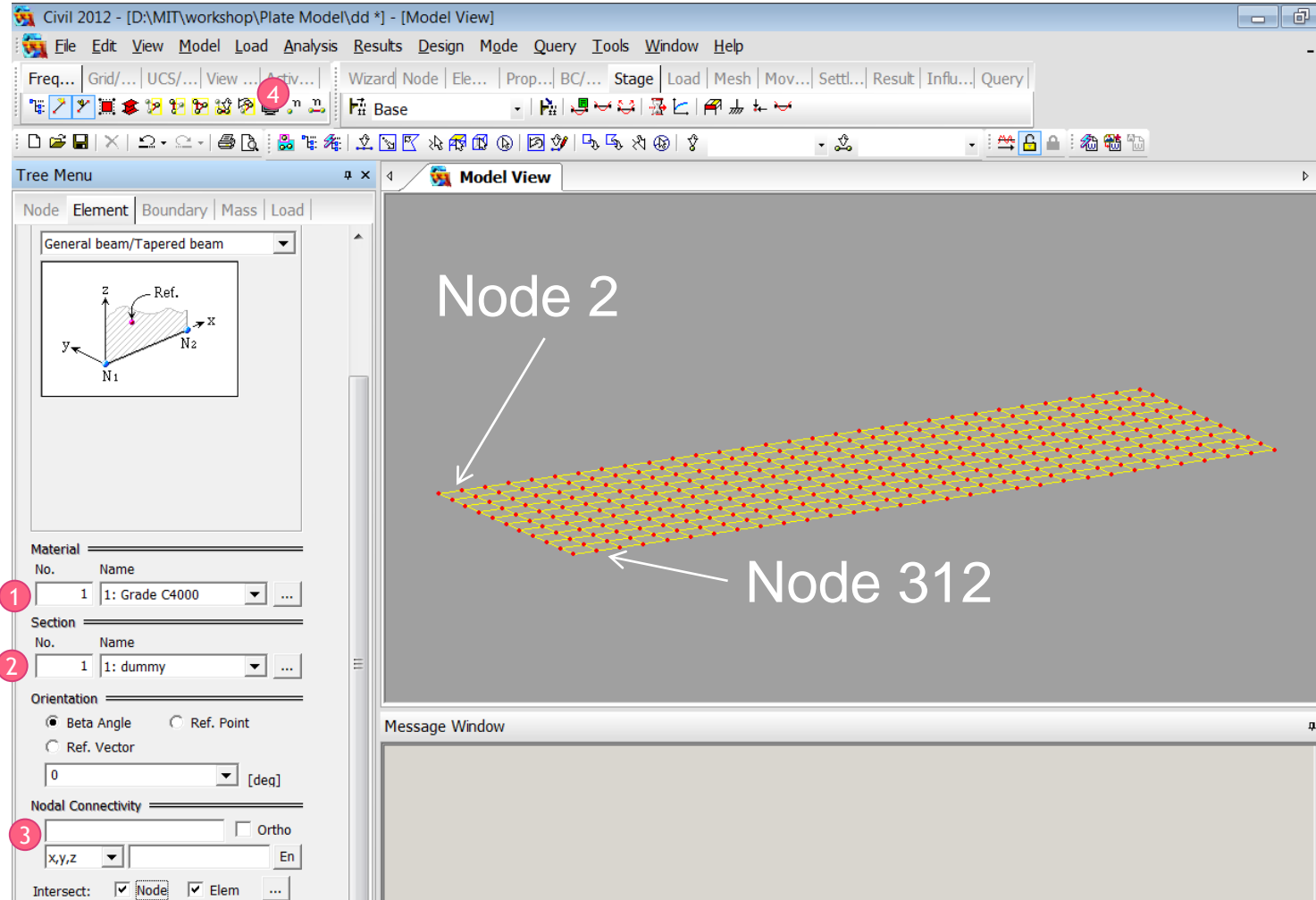
## ▶ 7. Dummy beam modeling

### Model → Elements →

#### Create elements

1. Material → 1: Grade c 4000
  2. Section 1: dummy
  3. Nodal connectivity
- Click on it and it will turn green after it turns green Click on nodes 2 and 312


Note: the node number can be obtained from 4



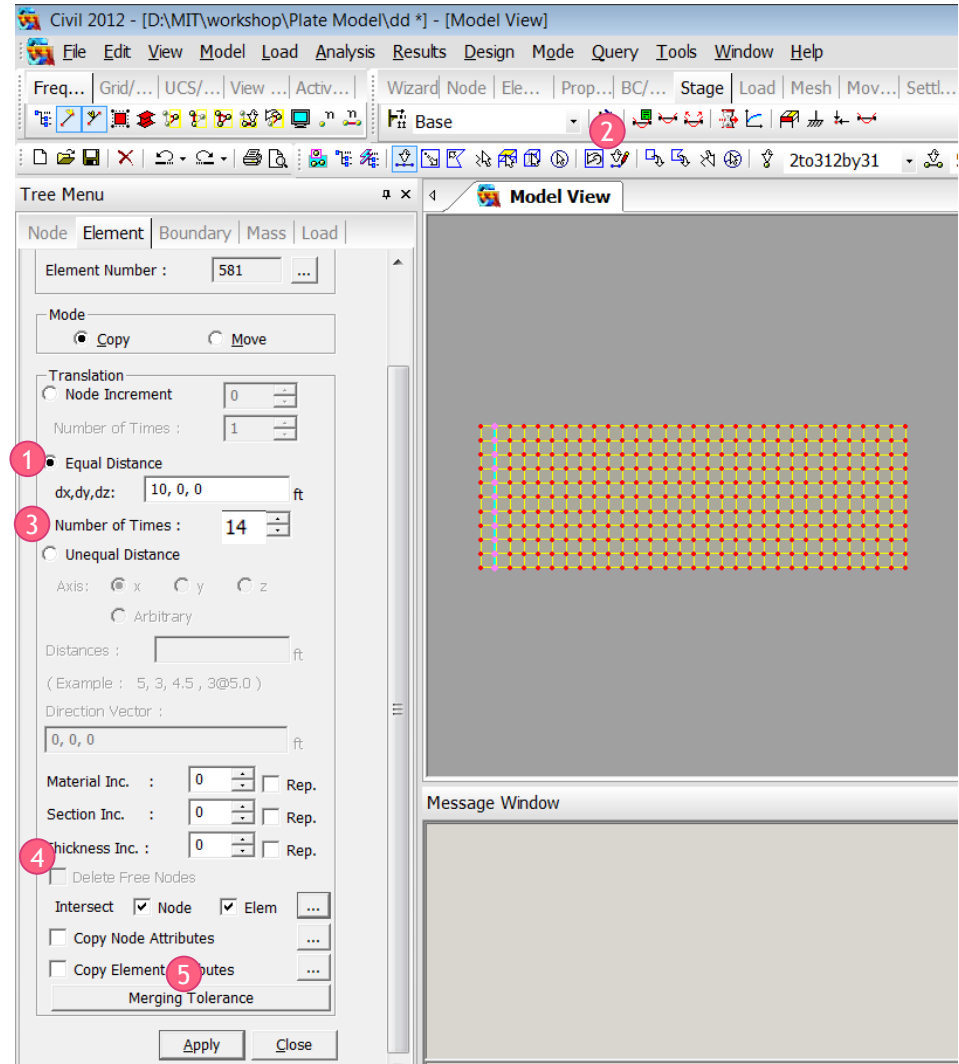
## ▶ 7. Dummy beam modeling

### Model → Elements →

#### Translate

1. Dx,dy,dz → 10,0,0
2. Click on  to select recently created beams
3. Number of Times → 14
4. Check on Intersect node and element
5. Click → Apply

Note: the node number can be obtained from 4





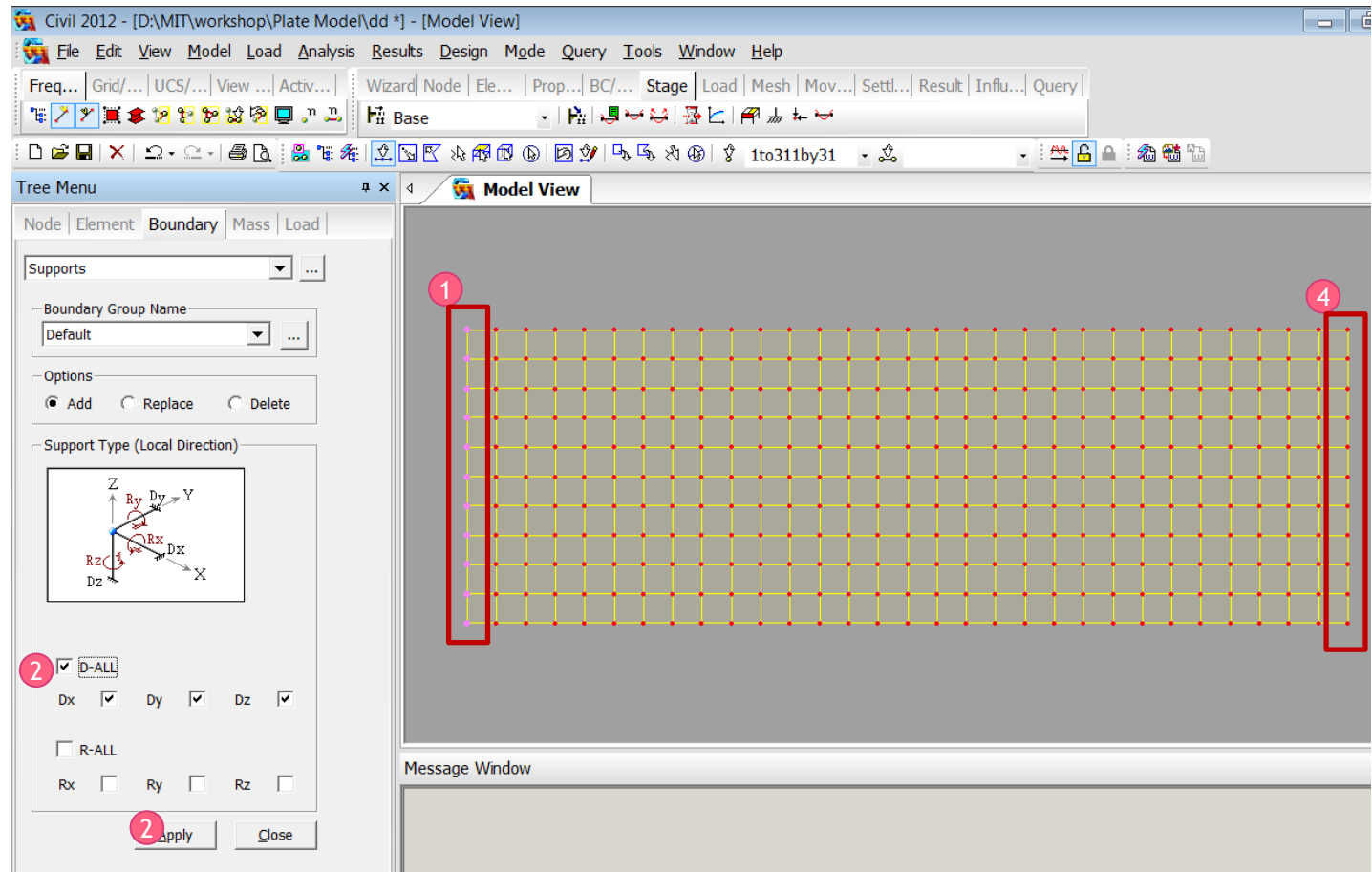
## ▶ 8. Support definition

### Defining sections for dummy beams

#### Model → Boundaries → Supports

1. Select Nodes as shown for 1
2. Select D all
3. Click → Apply
4. Select Nodes as shown in 4
5. Select Dy and Dz
6. Click → Apply

Note: the node number can be obtained from 4



## ▶ 9. Load Definition

### Defining sections for dummy beams

#### Load → Static Load

##### Cases

1. Name → self weight
2. Type → Dead Load
3. Click → Add
4. Name → Prestressing
5. Type → Prestressing
6. Click → Add

Static Load Cases

1 Name : self weight

Case : All Load Case

2 Type : Dead Load (D)

Description :

3 Add

Modify

Delete

No	Name	Type	Description
1	self weigh	Dead Load (D)	
2	prestressi	Prestress (PS)	
*			

Close

## ▶ 9. Load Definition

### Load → Self weight

1. Load Case Name → Self Weight
2. Z → -1
3. Click → Add

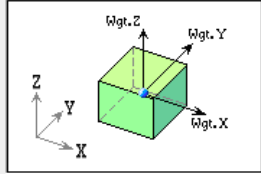
Node | Element | Boundary | Mass | **Load**

Self Weight

Load Case Name  
1 self weight

Load Group Name  
Default

Self Weight Factor



X 0

Y 0

2 Z -1

Load Case	X	Y	Z	Group
self weight	0	0	-1	Default


Operation

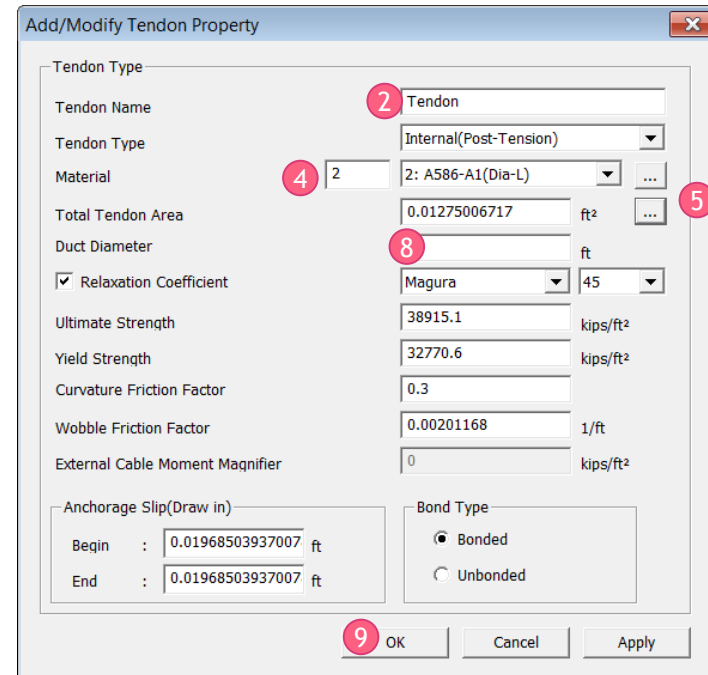
3 Add Modify Delete

Close

## ▶ 10. Tendon definition

### Load → Prestress Loads → Tendon Profile

1. Click Add
2. Name → Tendon L1
3. Tendon Type → Internal ( POST –Tensioned)
4. Material → 2: A586-A1(Dia-L)
5. Tendon Total Area → Click on 
6. Number of Strands → 12
7. Click → Ok
8. Duct Diameter → .3
9. Click → OK



**Add/Modify Tendon Property**

Tendon Name: Tendon

Tendon Type: Internal(Post-Tension)

Material: 2: A586-A1(Dia-L)

Total Tendon Area: 0.01275006717 ft²

Duct Diameter: 0.3 ft

☒ Relaxation Coefficient: Magura 45

Ultimate Strength: 38915.1 kips/ft²

Yield Strength: 32770.6 kips/ft²

Curvature Friction Factor: 0.3

Wobble Friction Factor: 0.00201168 1/ft

External Cable Moment Magnifier: 0 kips/ft²

Anchorage Slip(Draw in):

Begin : 0.01968503937007 ft

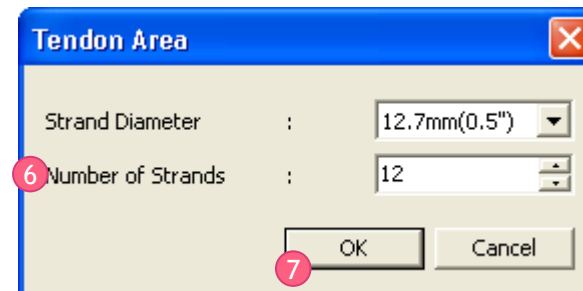
End : 0.01968503937007 ft

Bond Type:

☒ Bonded

☐ Unbonded

OK Cancel Apply



**Tendon Area**

Strand Diameter : 12.7mm(0.5")

Number of Strands : 12

OK Cancel

## ▶ 10. Tendon definition

### Load → Prestress Loads → Tendon Profile

1. Click Add
2. Name → Tendon L1
3. Tendon Property → Tendon
4. Assigned Elements → 301 to 330
5. Input Type → 3D
6. 0,0,0 and 150,0,0
7. Profile Insertion point → 301
8. Click → OK

**Add/Modify Tendon Profile**

2 Tendon Name : Tendon L1 Group : Default ...

3 Tendon Property : Tendon ...

4 Assigned Elements : 301to330

5 Input Type : ☐ 2-D ☒ 3-D

Curve Type : ☒ Spline ☐ Round

Straight Length of Tendon  
Begin : 0 ft  
End : 0 ft

☐ Typical Tendon No. of Tendons : 1

Transfer Length  
User defined Length Begin : 0 End : 0 ft

Profile  
Reference Axis : ☐ Straight ☐ Curve ☒ Element

Y 8.46154  
-11.5385  
0 20 40 60 80 100 130 x

Z 8.46154  
-11.5385  
0 20 40 60 80 100 130 x

	x(ft)	y(ft)	z(ft)	fix	Ry[deg]	Rz[deg]
1	0.0000	0.0000	0.0000	<input type="checkbox"/>	0.00	0.00
2	150.00	0.0000	0.0000	<input type="checkbox"/>	0.00	0.00
3				<input type="checkbox"/>		

6

Point of Sym.: ☐ First ☒ Last Make Symmetric Tendon

7 Profile Insertion Point : ☒ End-I ☐ End-J of Elem. 301

X Axis Direction : ☒ I->J ☐ J->I of Elem. 301

X Axis Rot. Angle : 0 [deg] ☒ Projection

Offset y : 0 ft z : 0 ft

OK Cancel Apply

# ▶ 10. Tendon definition

## Load → Prestress Loads → Tendon Profile

1. Select Tendon L1
  2. Click →copy /move
  3. Select → new Assigned elements
  4. Assigned Elements → 331 to 360
  5. Insertion element → 331
  6. Click →Add
  7. Similarly add elements as shown and then hit ok
- \*8 new tendons will be defined

**Copy/Move Tendon Profile**

Mode  
☒ Copy ☐ Move

Translation  
☒ Element Increment

☐ Equal Distance  
dx,dy,dz:  ft

☐ Current Assigned Elements

☒ New Assigned Element

No	Assigned	Insertion
1	331to360	331
2	361to390	361
3	391to420	391
4	421to450	421
5	451to480	451
6	481to510	481
7	511to540	511
8	541to570	541
*		

Assigned Elements :

Insertion Element :

☒ Add

☐ Auto-Adjustment of Tendon Length

OK Cancel

# 10. Tendon definition

## Load → Prestress Loads → Tendon Profile

1. Click Add
2. Name → Tendon T1
3. Tendon Property → Tendon
4. Assigned Elements → 571 to 580
5. Input Type → 3D
6. 0,0,0 and 50,0,1
7. Profile Insertion point → 571
8. Click → OK

**Add/Modify Tendon Profile**

2 Tendon Name : Tendon T1 Group : Default ...

3 Tendon Property : Tendon ...

4 Assigned Elements : 571to580

5 Input Type : ☐ 2-D ☒ 3-D

Curve Type : ☒ Spline ☐ Round

Straight Length of Tendon : Begin : 0 ft End : 0 ft

☐ Typical Tendon No. of Tendons : 1

Transfer Length : User defined Length Begin : 0 End : 0 ft

Profile : Reference Axis : ☐ Straight ☐ Curve ☒ Element

Y : 1.15385 -3.84615 x

Z : 1.15385 -3.84615 x

	x(ft)	y(ft)	z(ft)	fix	Ry[deg]	Rz[deg]
1	0.0000	0.0000	1.0000	<input type="checkbox"/>	0.00	0.00
2	50.000	0.0000	1.0000	<input type="checkbox"/>	0.00	0.00
3				<input type="checkbox"/>		

6

Point of Sym.: ☐ First ☒ Last Make Symmetric Tendon

7 Profile Insertion Point : ☒ End-I ☐ End-J of Elem. 571

x Axis Direction : ☒ I->J ☐ J->I of Elem. 571

x Axis Rot. Angle : 0 [deg] ☒ Projection

Offset y : 0 ft z : 0 ft

8 OK Cancel Apply

## ▶ 10. Tendon definition

### Load → Prestress Loads → Tendon Profile

1. Select Tendon T1
2. Click →copy /move
3. Select → new Assigned elements
4. Assigned Elements → 331 to 360
5. Insertion element → 331
6. Click →Add
7. Similarly add elements as shown and then hit ok

new tendons will be defined  
Go to works tree and display tendons there will be  
24 tendons

**Copy/Move Tendon Profile**

Mode  
☒ Copy ☐ Move

Translation  
☐ Element Increment     
☐ Equal Distance  
 dx,dy,dz:  ft  
☐ Current Assigned Elements

**3** ☒ New Assigned Element

	No	Assigned	Insertion
▶	1	581to590	581
	2	591to600	591
	3	601to610	601
	4	611to620	611
	5	621to630	621
	6	631to640	631
	7	641to650	641
	8	651to660	651
	9	661to670	661
	10	671to680	671

**4** Assigned Elements :

Insertion Element :

**5**

**6** ☐ Auto-Adjustment of Tendon Length

581to590	581
591to600	591
601to610	601
611to620	611
621to630	621
631to640	631
641to650	641
651to660	651
661to670	661
671to680	671
681to690	681
691to700	691
701to710	701
711to720	711



## ▶ 10. Tendon definition

**Load → Prestress Loads → Tendon Prestress Loads**

1. Load case → prestressing
2. Select all tendons
3. Begin and end → 28800
4. Click → Add

Analysis -> Perform Analysis

Node | Element | Boundary | Mass | Load

Tendon Prestress Loads

Load Case Name  
prestressing

Load Group Name  
Default

Select Tendon for Loading

Tendon	Selected
Name	Name
	Tendon L1
	Tendon L1-Cc
	Tendon L1-Cc
	Tendon L1-Cc
	Tendon L1-Cc
	Tendon L1-Cc

Stress Value

☒ Stress ☐ Force

1st Jacking : Begin

Begin : 28800 kips/ft<sup>2</sup>

End : 28800 kips/ft<sup>2</sup>

Grouting : after 0 Stage

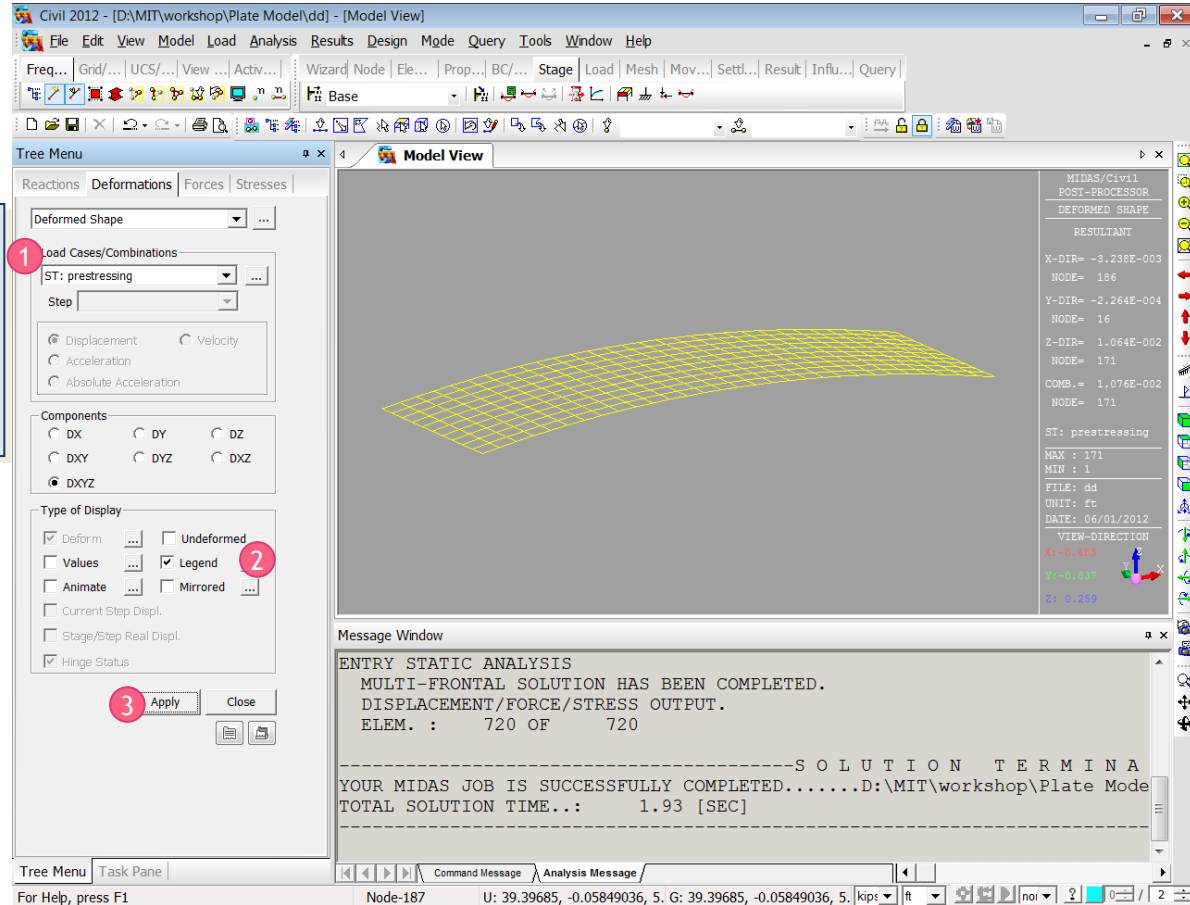
Tendon	Type	Load Case	Gr
--------	------	-----------	----

Add Modify Delete

# ► Results

## Results → Deformations

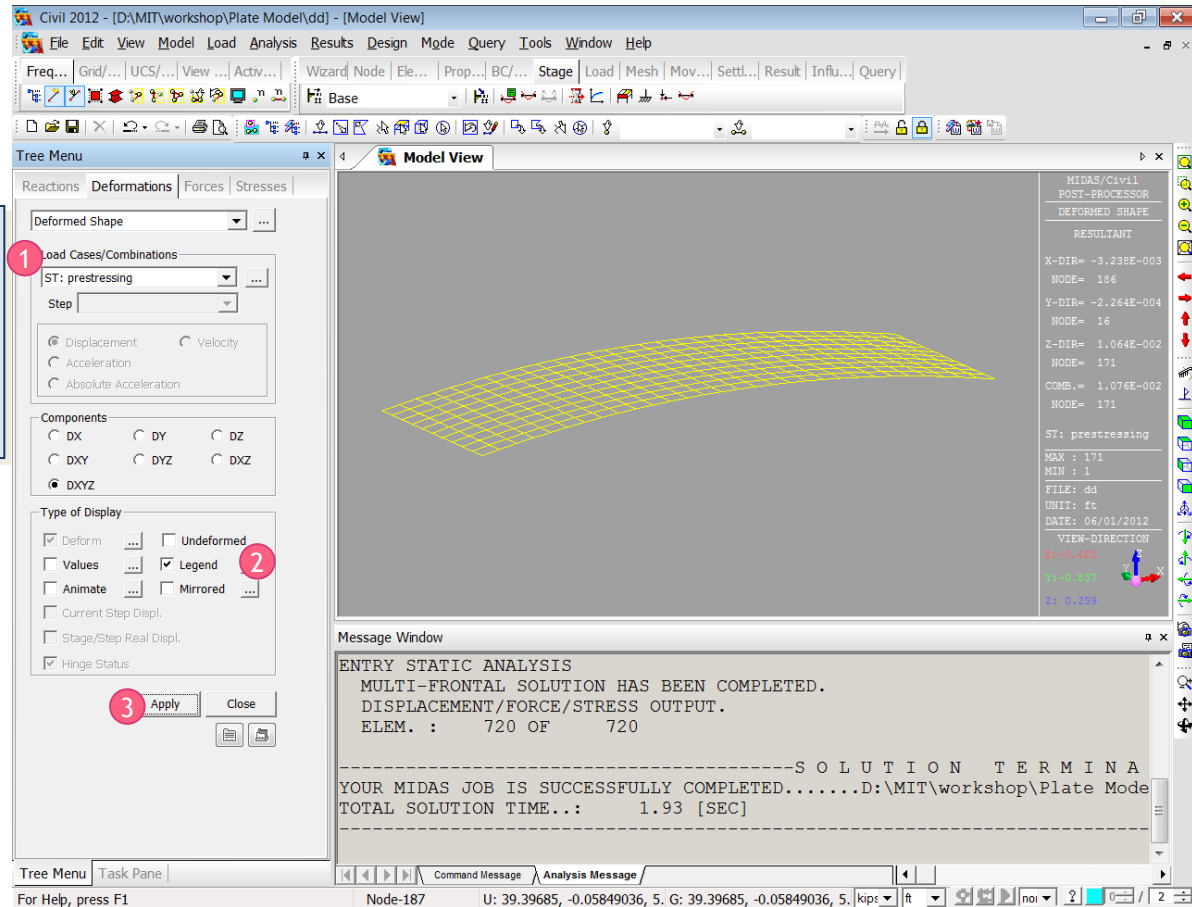
1. Load case → prestressing
2. Check on Legend
3. Click → Apply



# Results

## Results → Deformations

1. Load case → prestressing
2. Check on Legend
3. Click → Apply



# Thank You

Any question or suggestion:

Nithil Malguri

[nmalguri@midasit.com](mailto:nmalguri@midasit.com)

646 852 9284