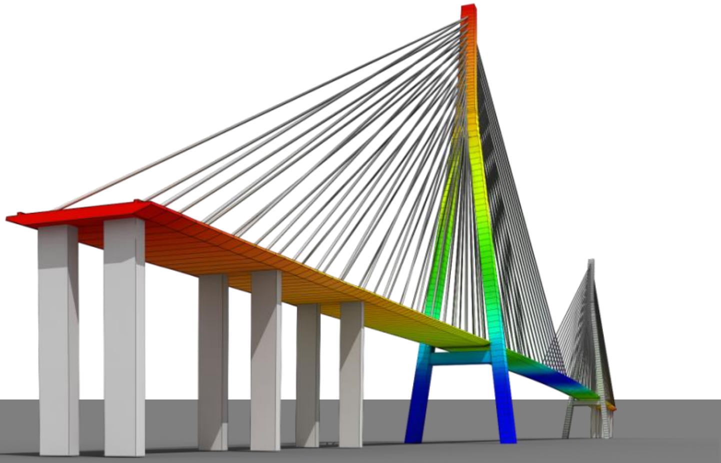
Prestressing Plate Elements
Midas Civil



Bridging Your Innovations to Realities

Introduction

Prestressing Plate Elements in Midas Civil.

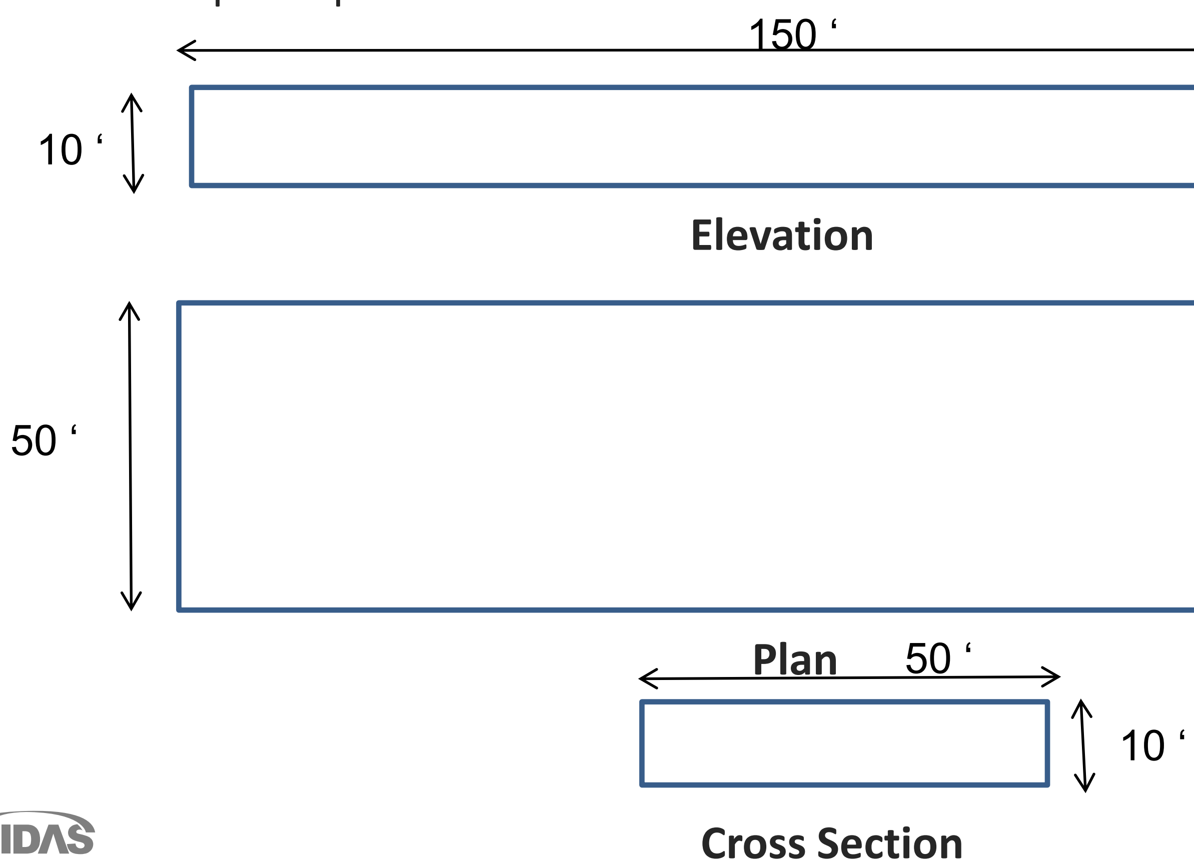
In Midas Civil the prestressing cannot be applied to the plate elements and hence the sole purpose of this tutorial is model the prestressing in the plate elements through a workaround.

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

Introduction

Bridge Introduction

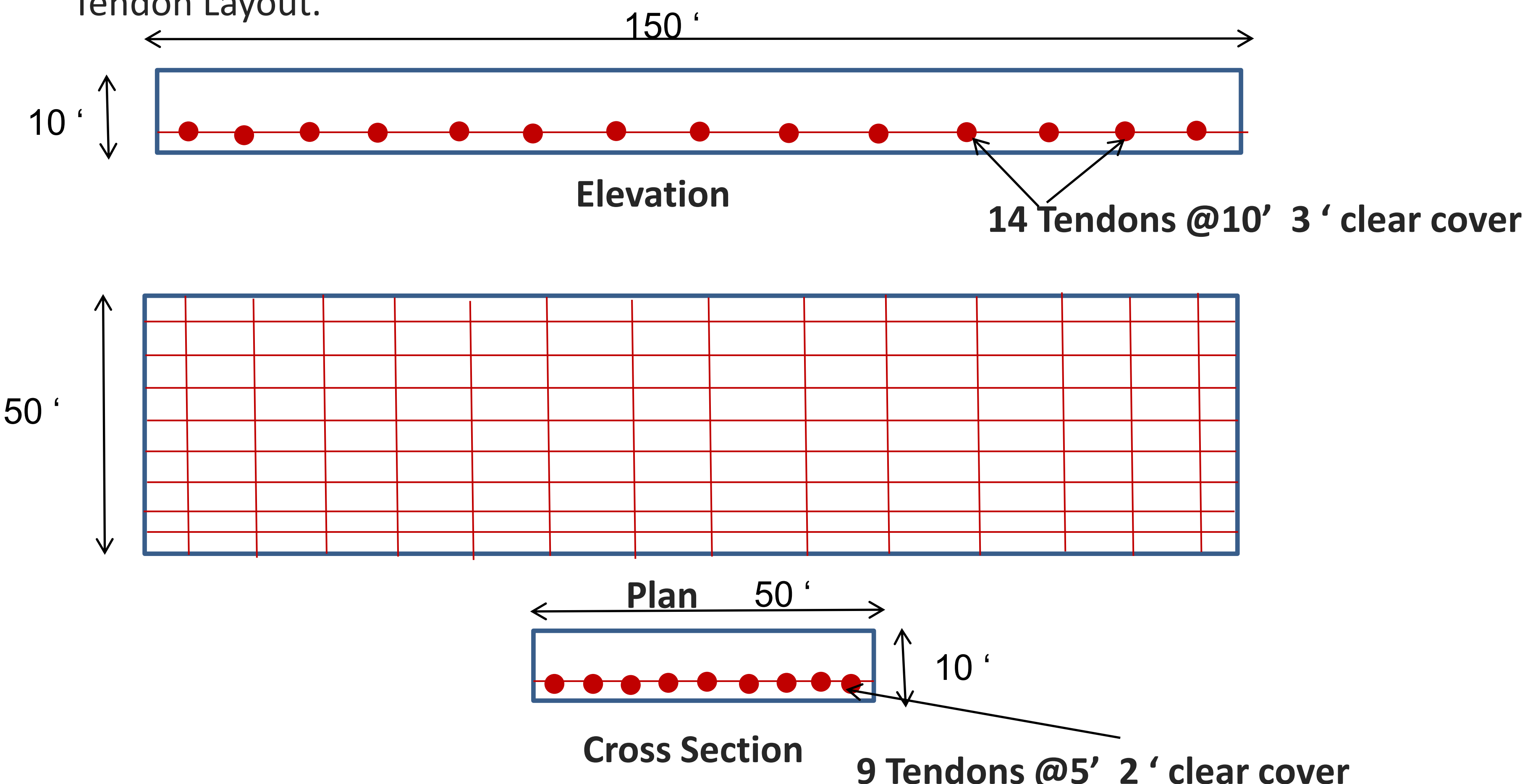
The following are the drawings of the bridge. For this tutorial the drawings are kept simple so that the user can understand the modeling technique.



Introduction

Bridge Introduction

Tendon Layout.



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.

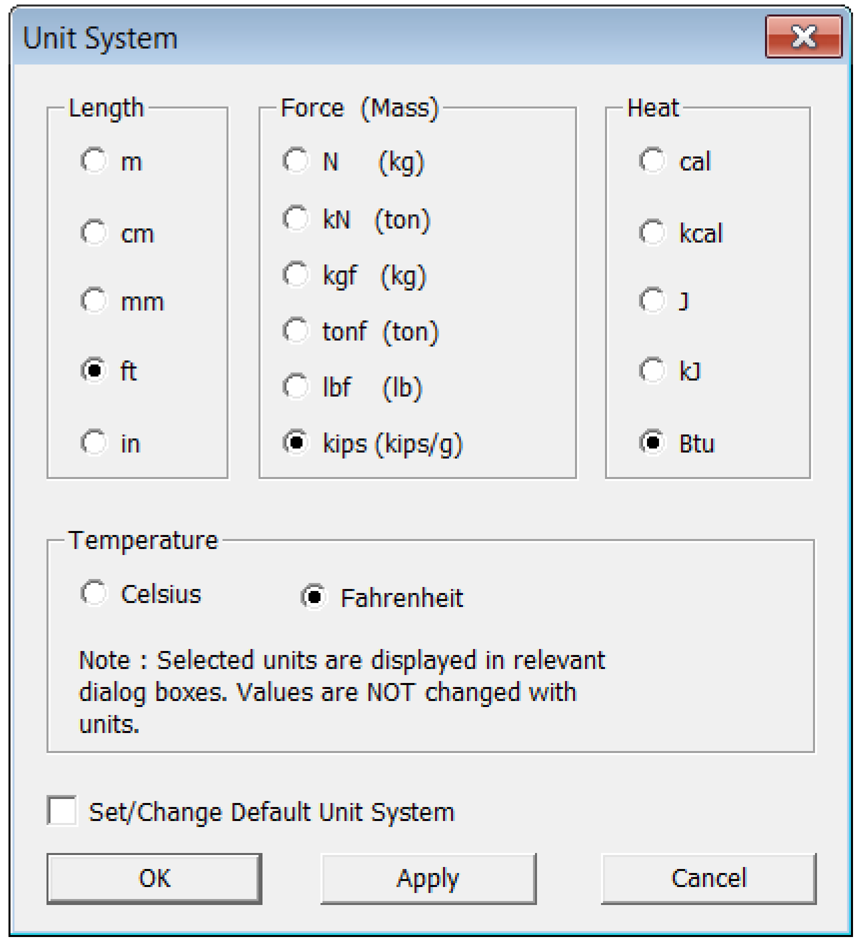
2. Setting Units

Setting of Units:

File → New Project

Tools → Unit System

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

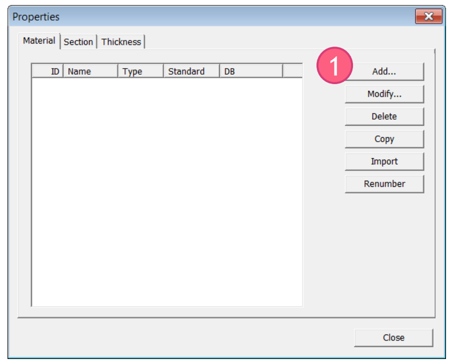


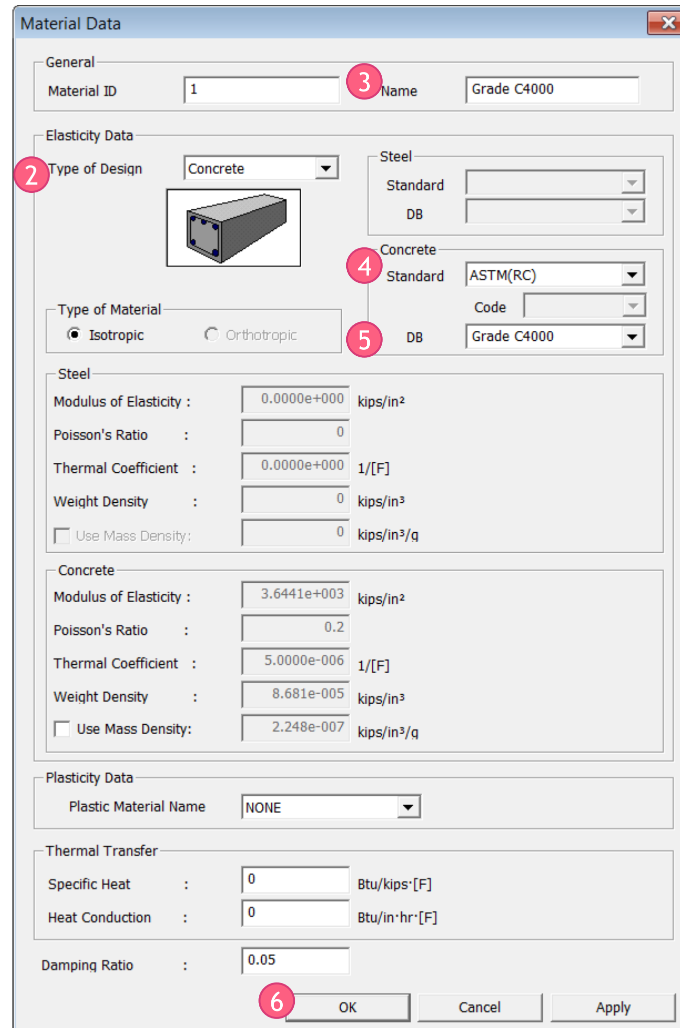
3. Material Definition

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





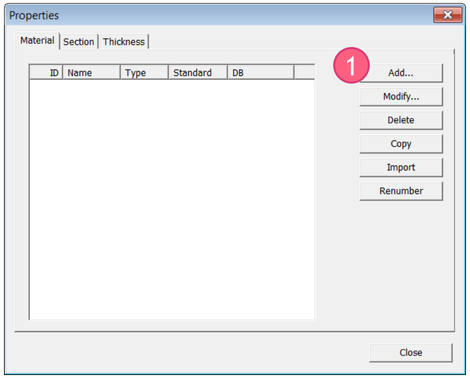
MIDAS

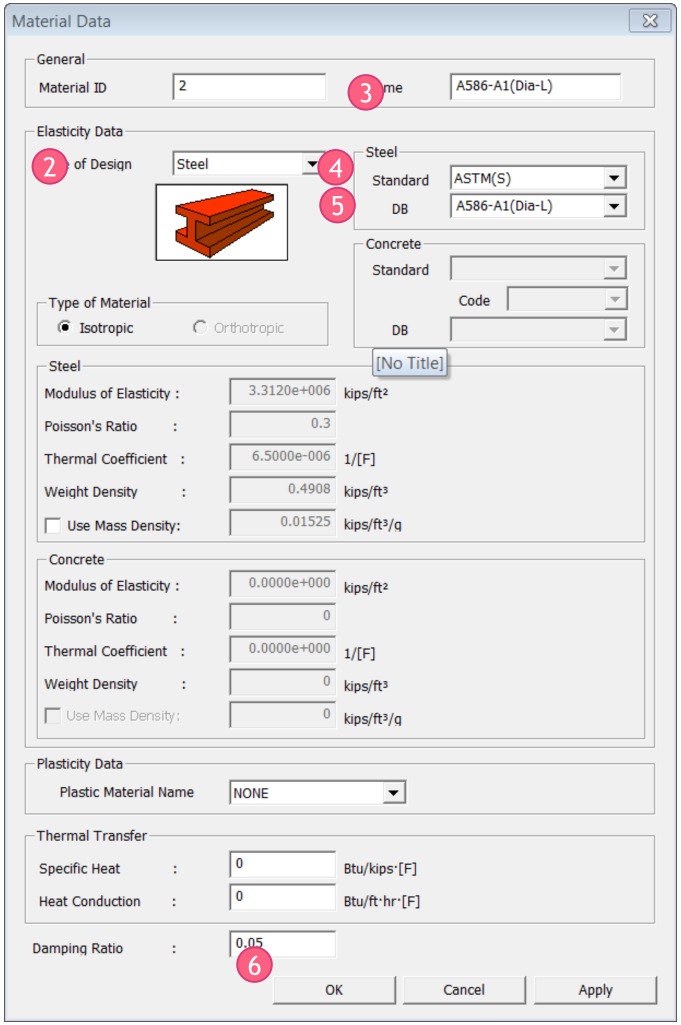
3. Material Definition

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



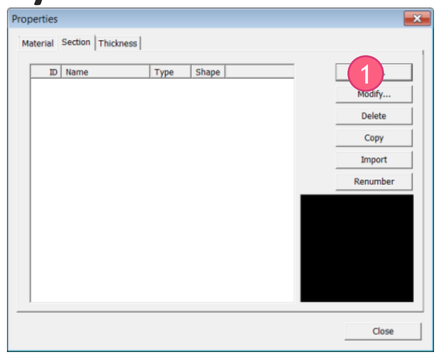


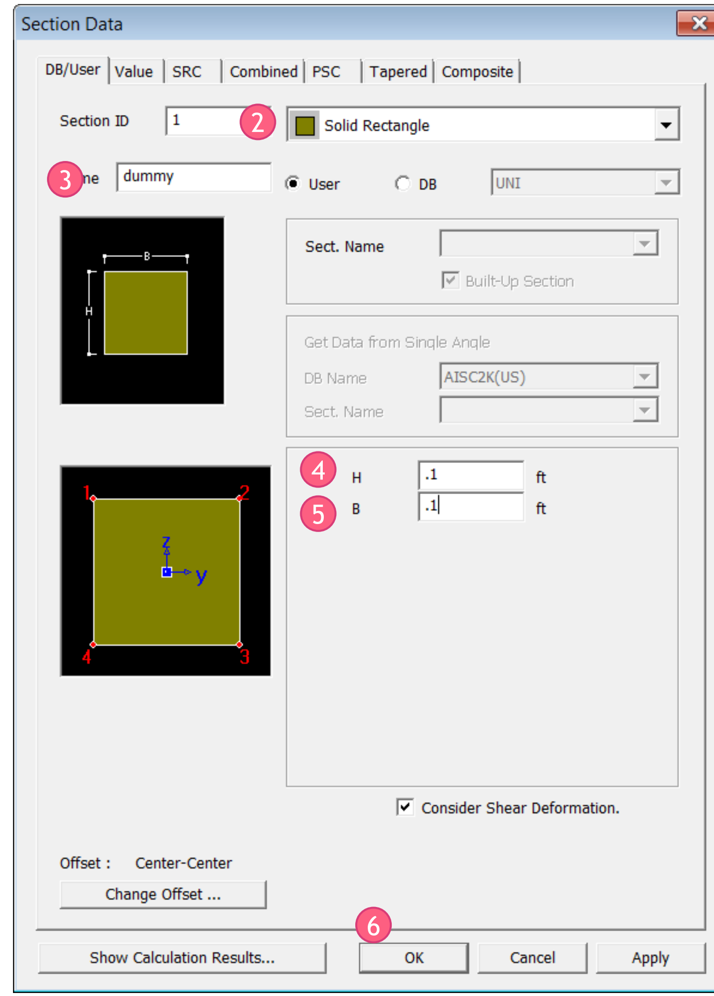
4. Section Definition

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

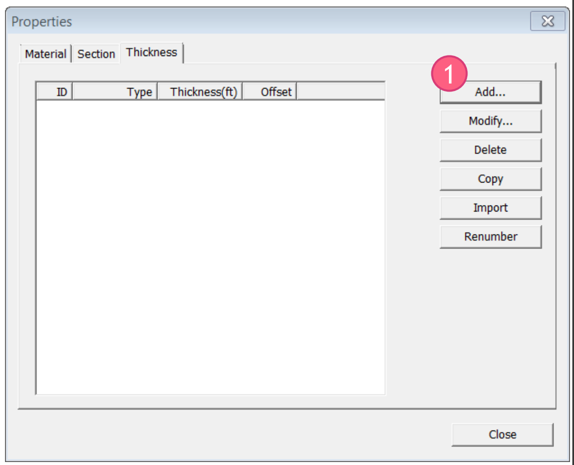


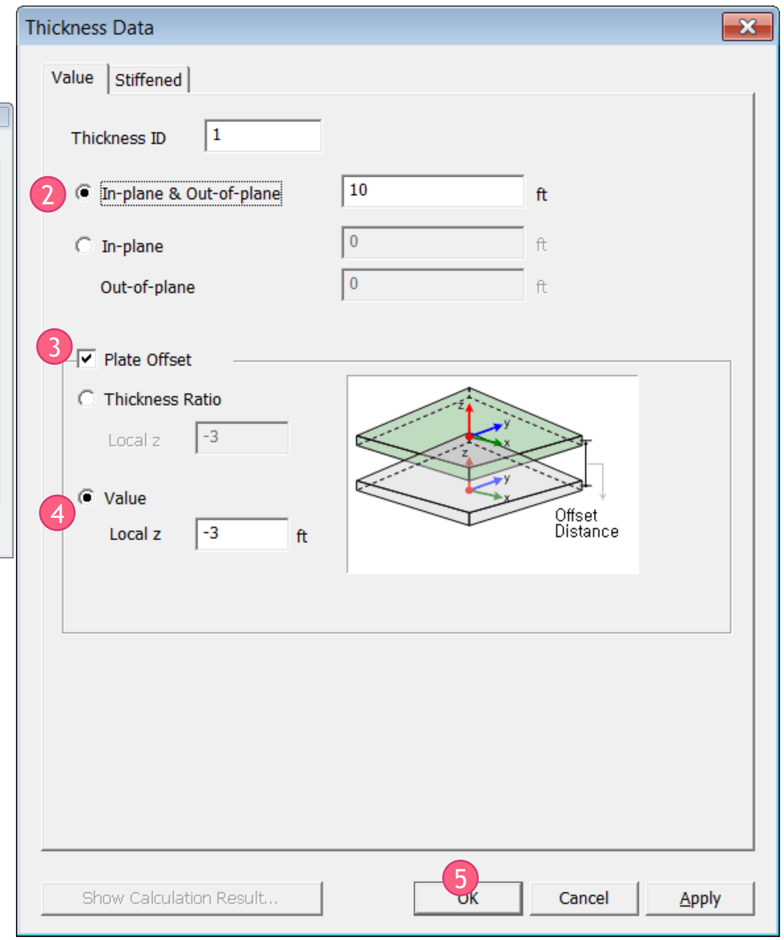


5. Thickness Definition

Model→ Properties→ Thickness

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

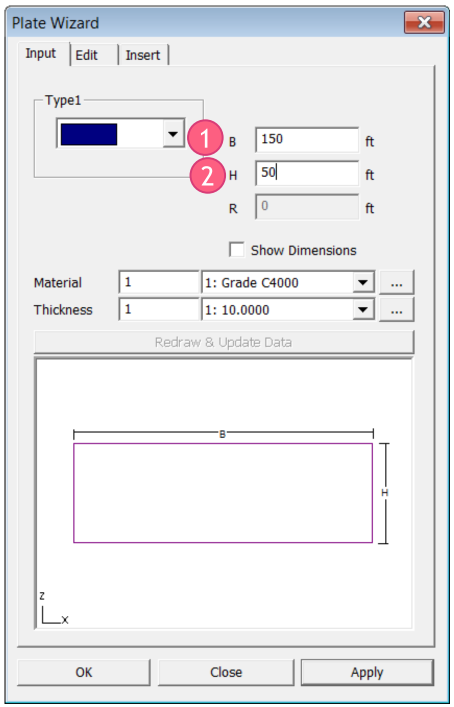


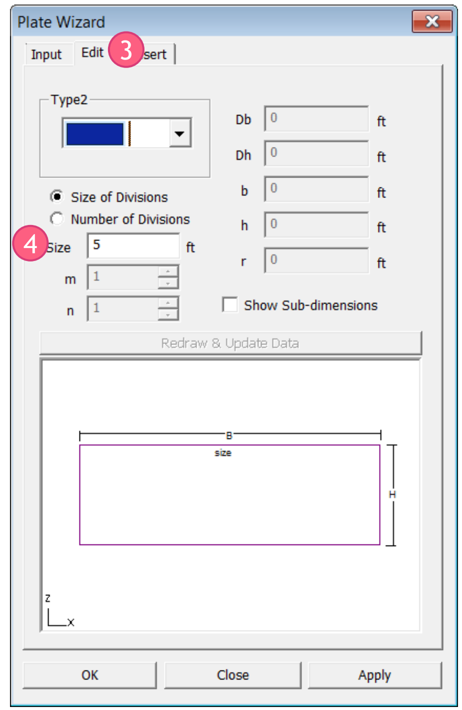


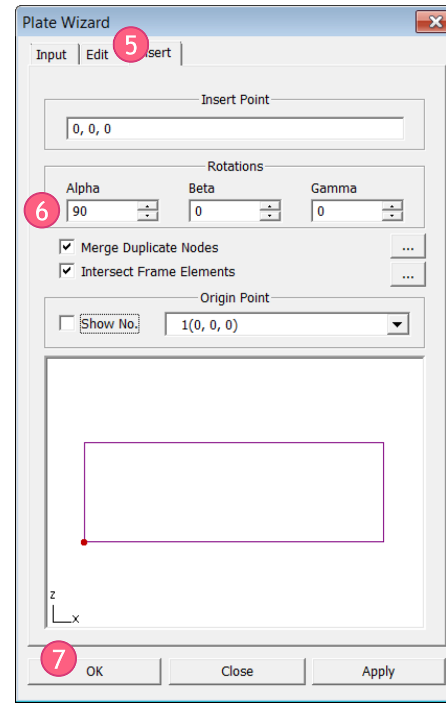
6. Slab Modeling

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







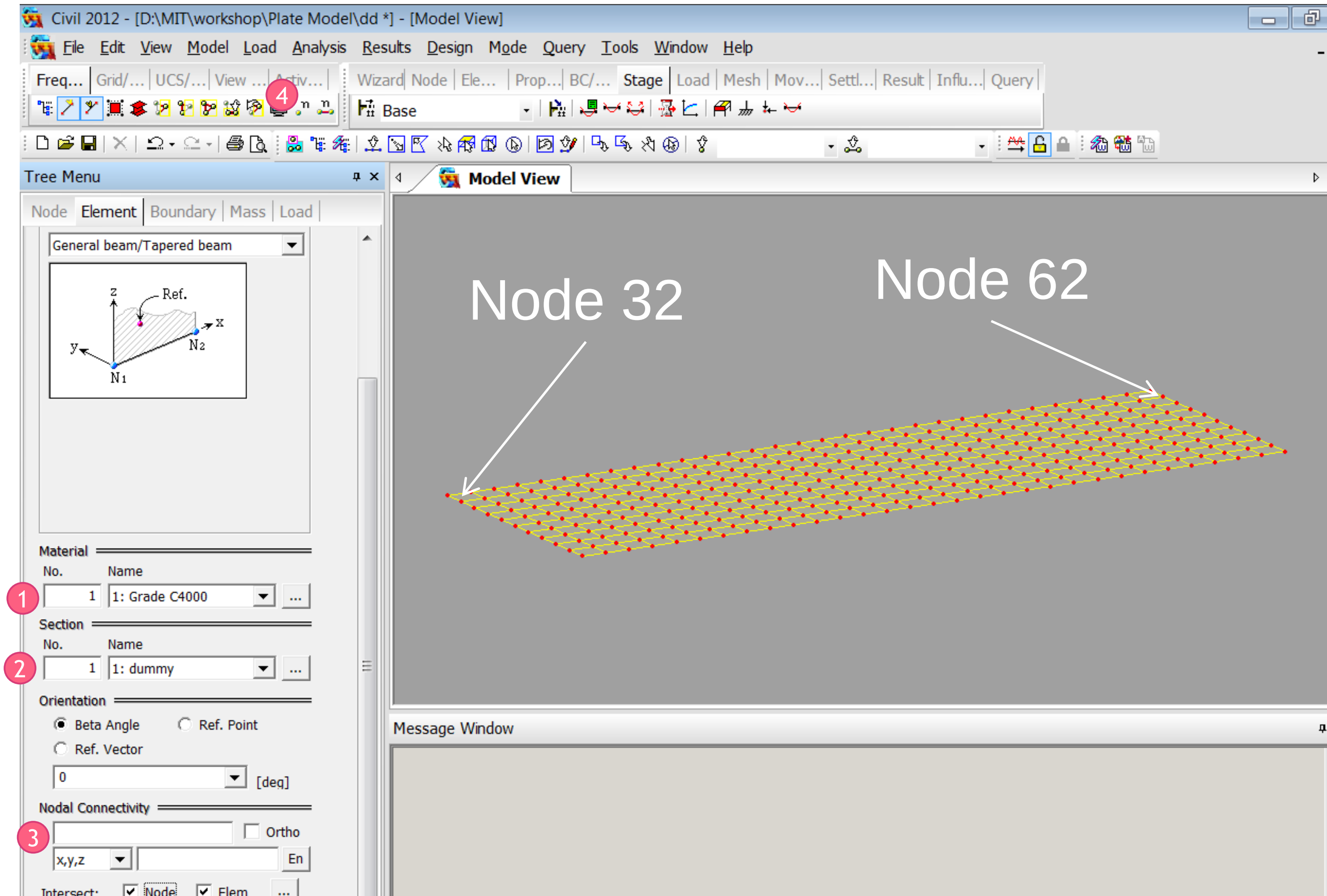
7. Dummy beam modeling

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

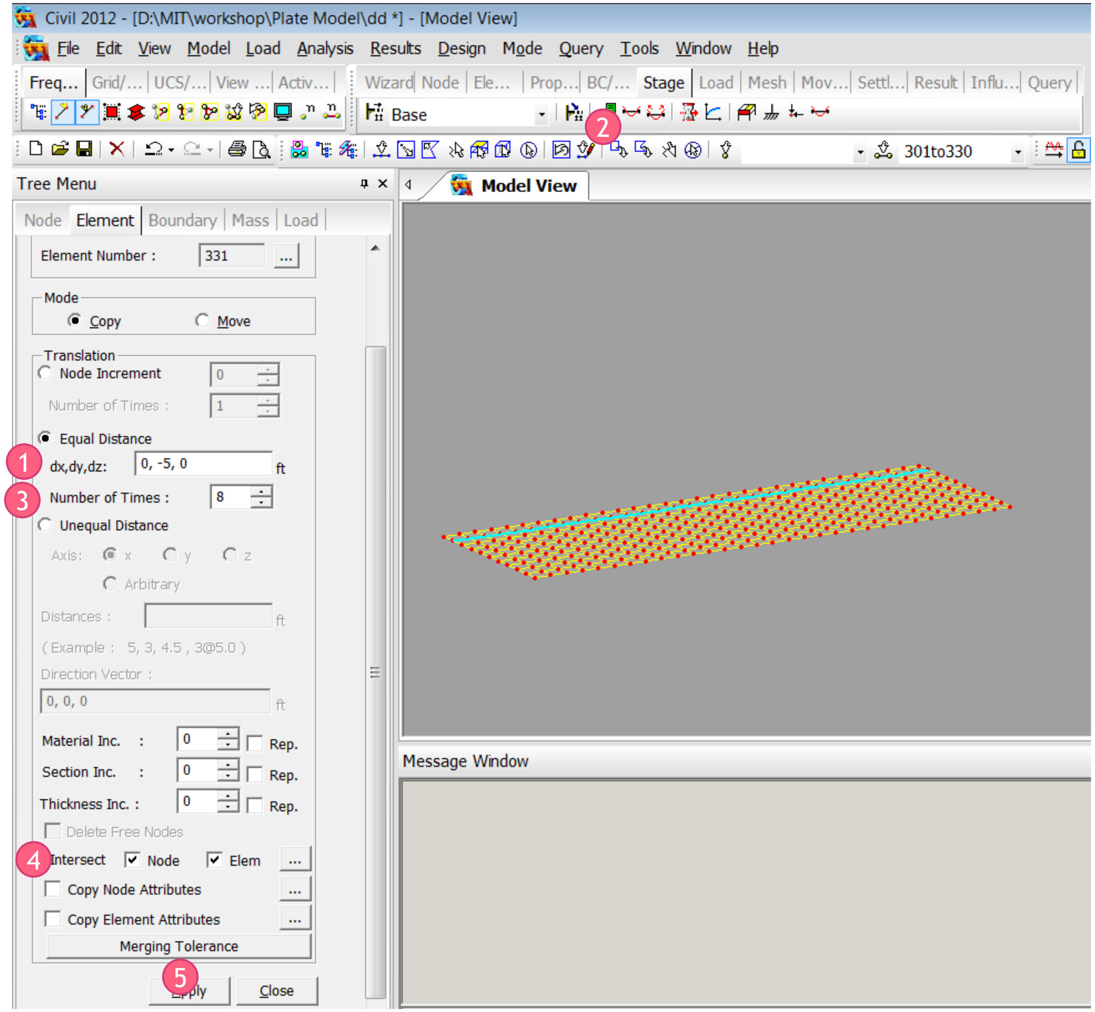


▶ 7. Dummy beam modeling

Model→ Elements → Translate

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

Note: the node number can be obtained from 4



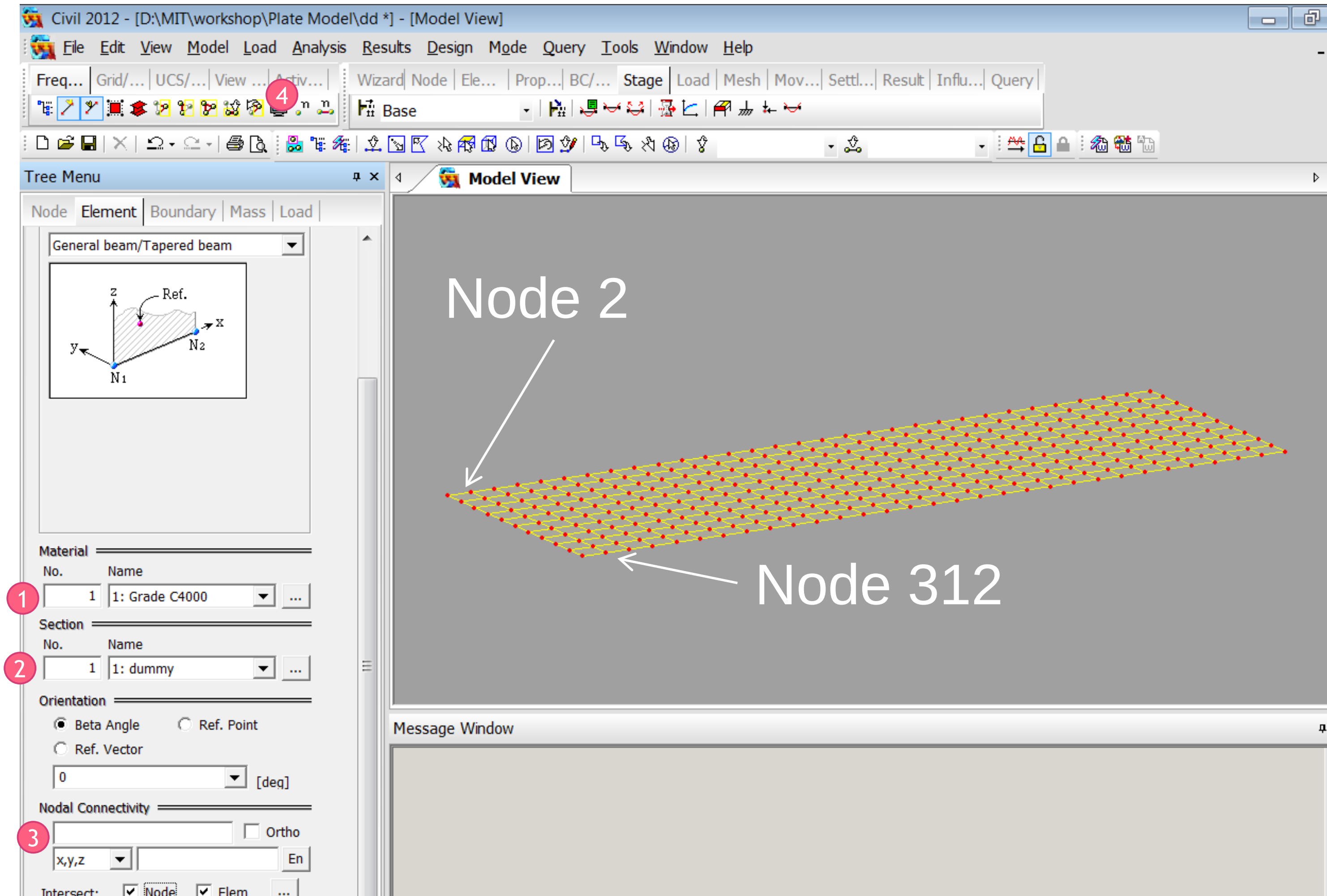
7. Dummy beam modeling

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

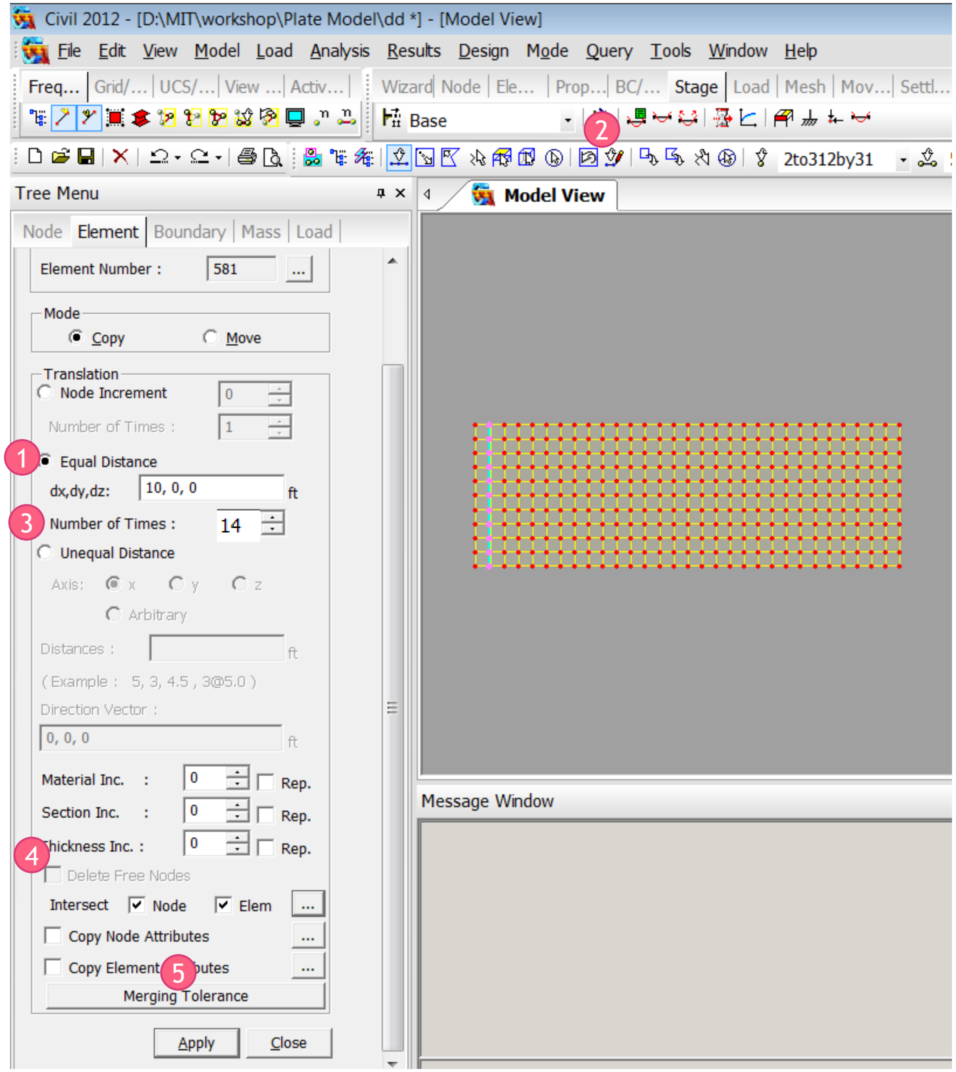


▶ 7. Dummy beam modeling

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



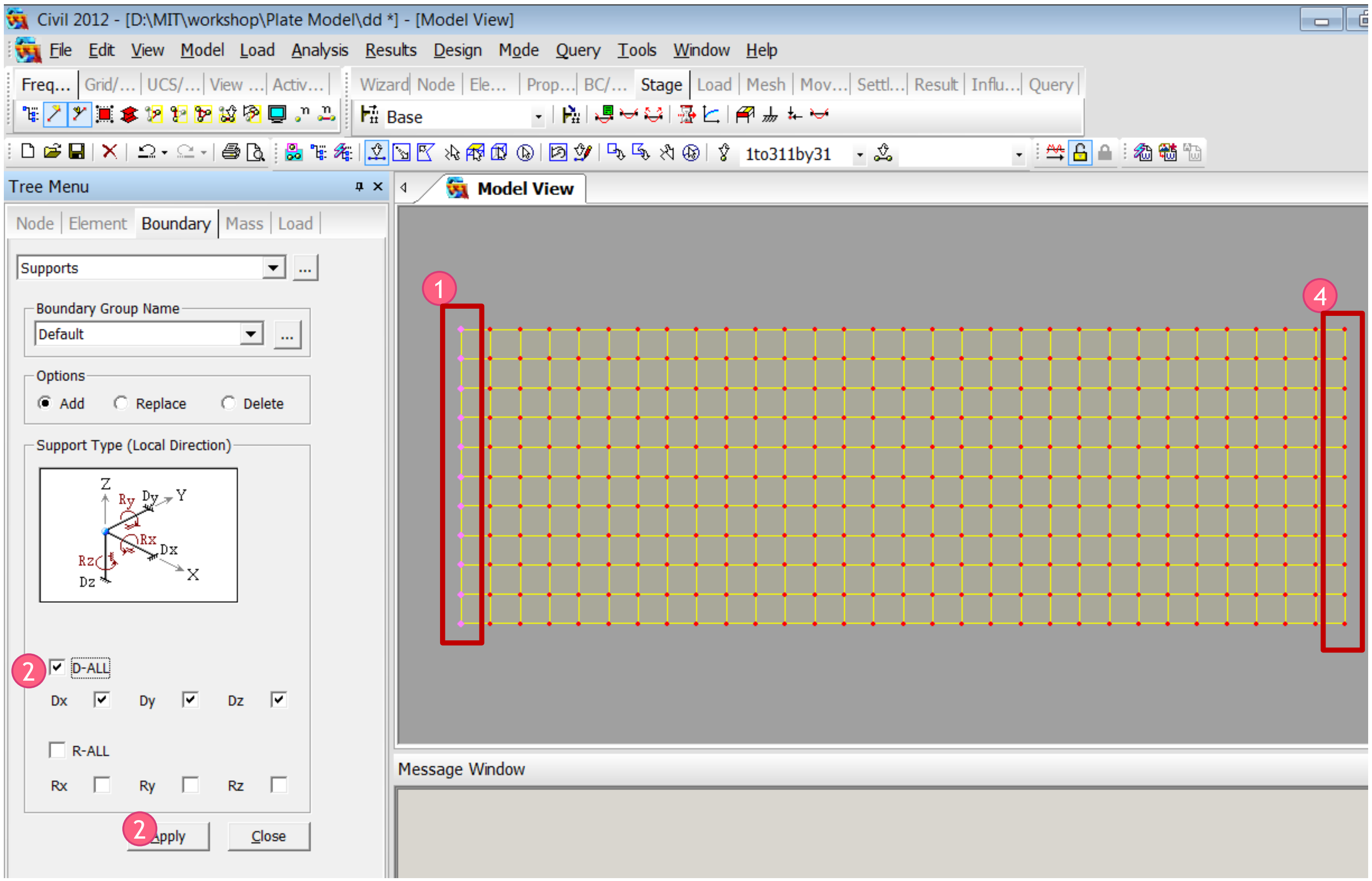
8. Support definition

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

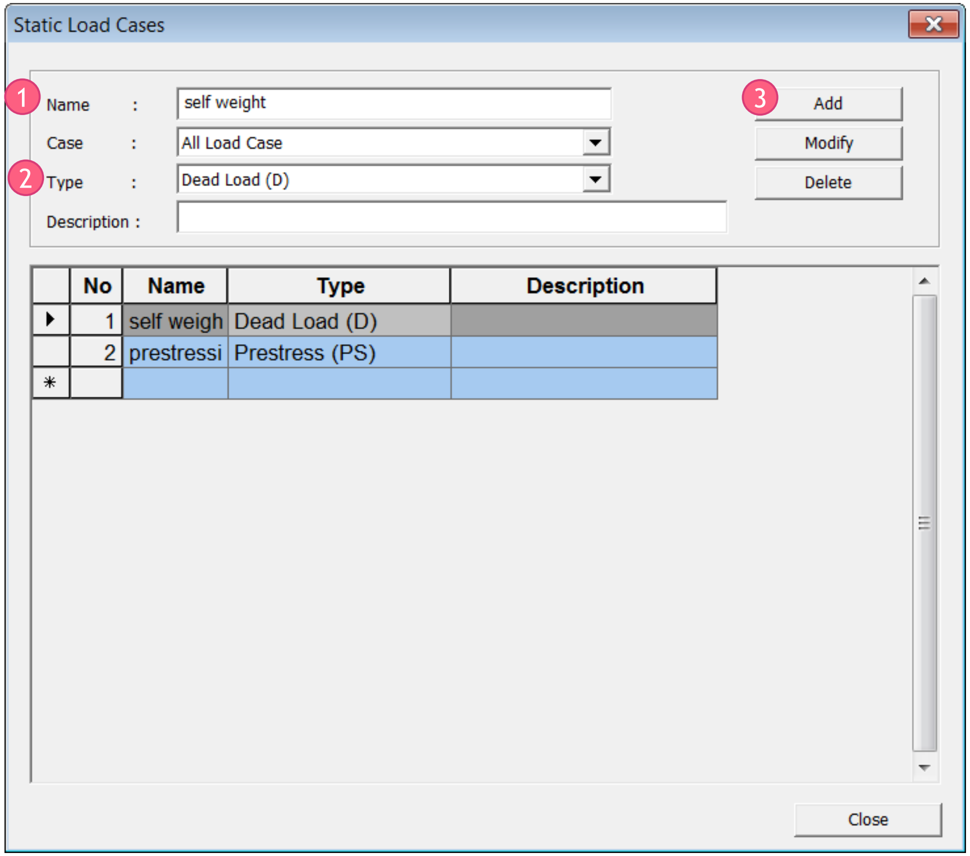


9. Load Definition

Defining sections for dummy beams

Load → Static Load Cases

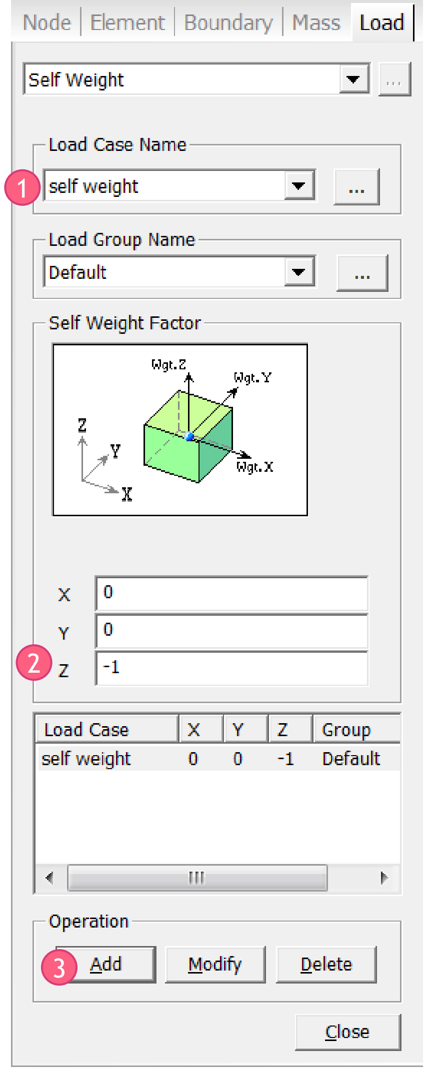
1. Name → self weight
2. Type → Dead Load
3. Click → Add
4. Name → Prestressing
5. Type → Prestressing
6. Click → Add



9. Load Definition

Load → Self weight

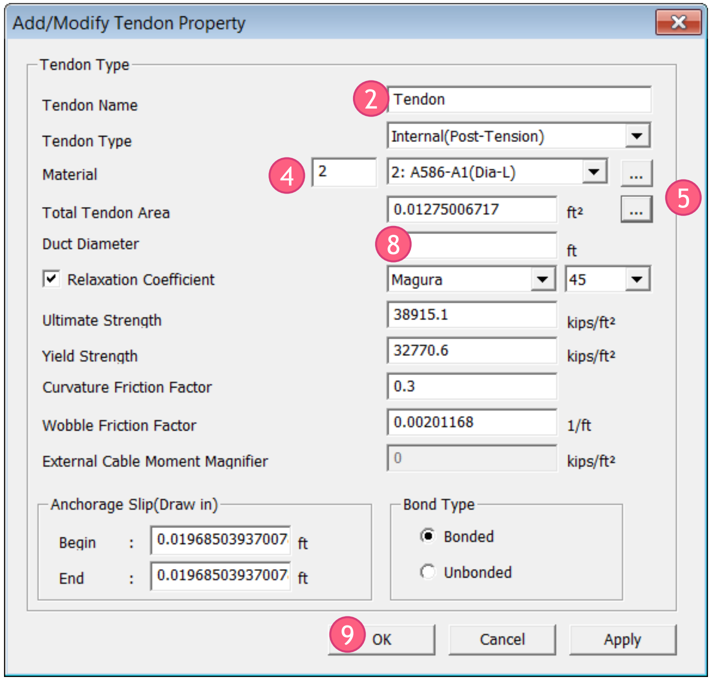
1. Load Case Name → Self Weight
2. Z → -1
3. Click →Add

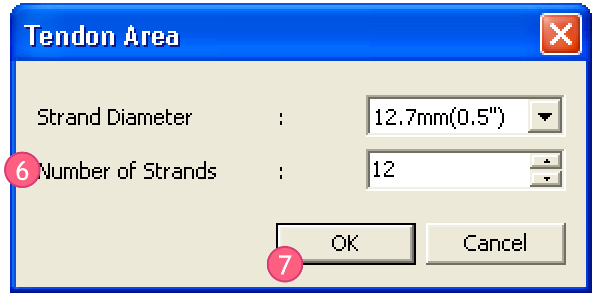


10. Tendon definition

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

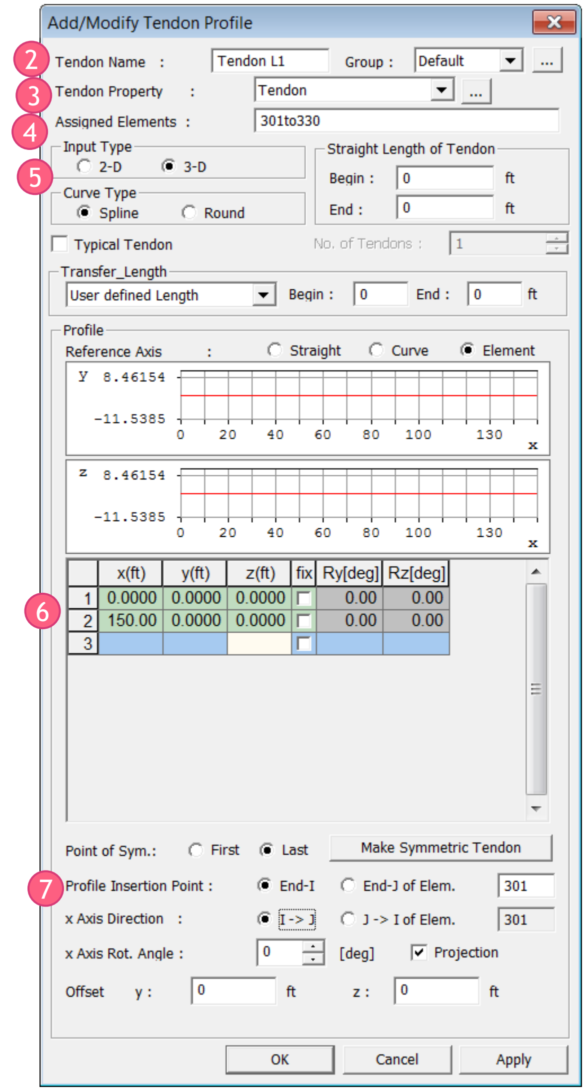




10. Tendon definition

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

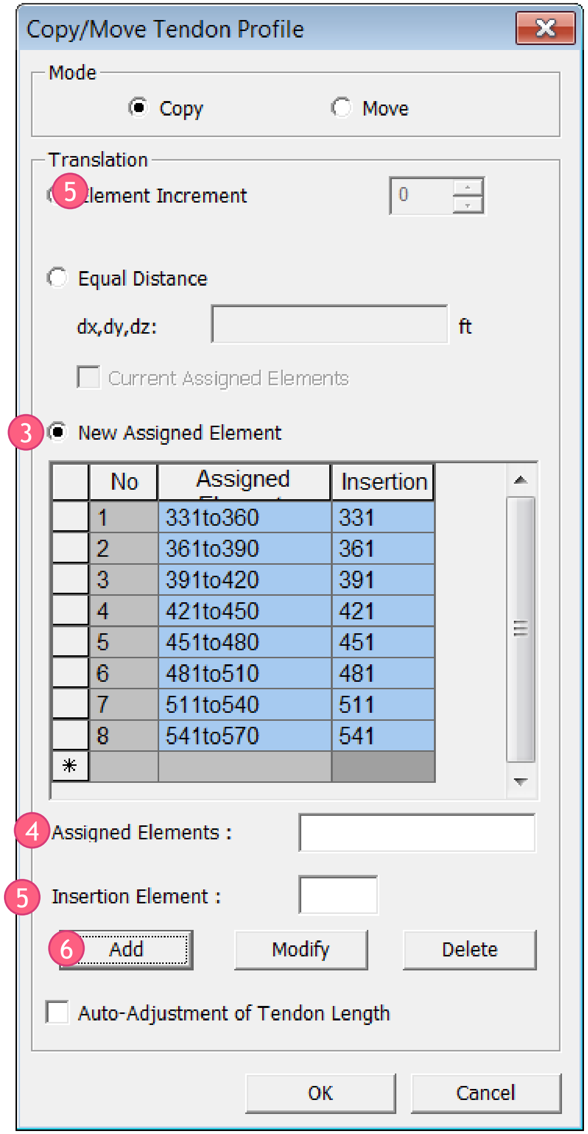


10. Tendon definition

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

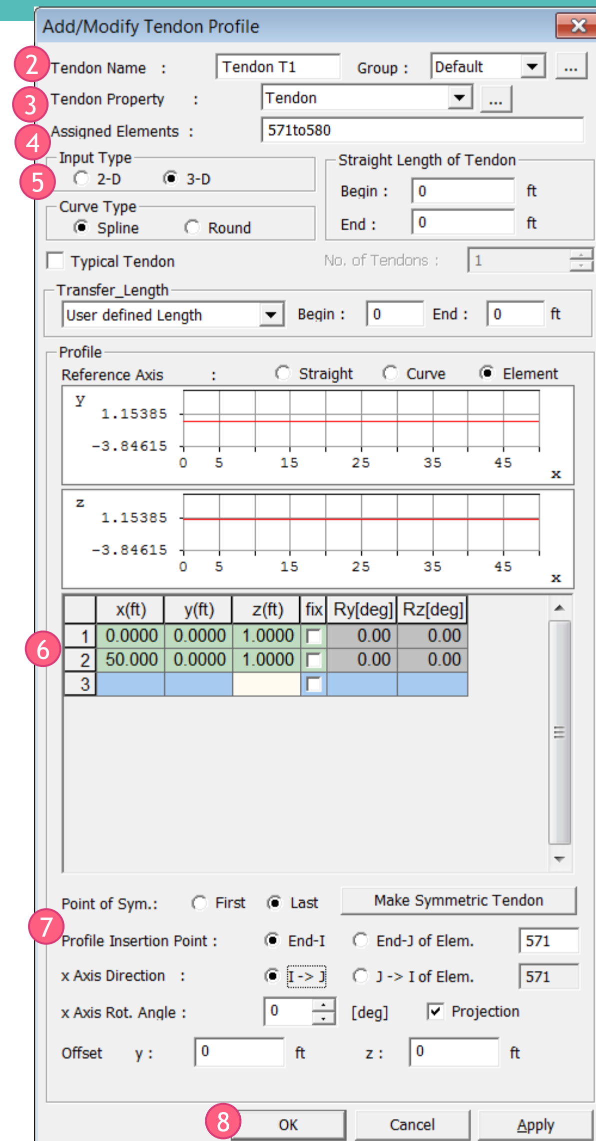
*8 new tendons will be defined



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



10. Tendon definition

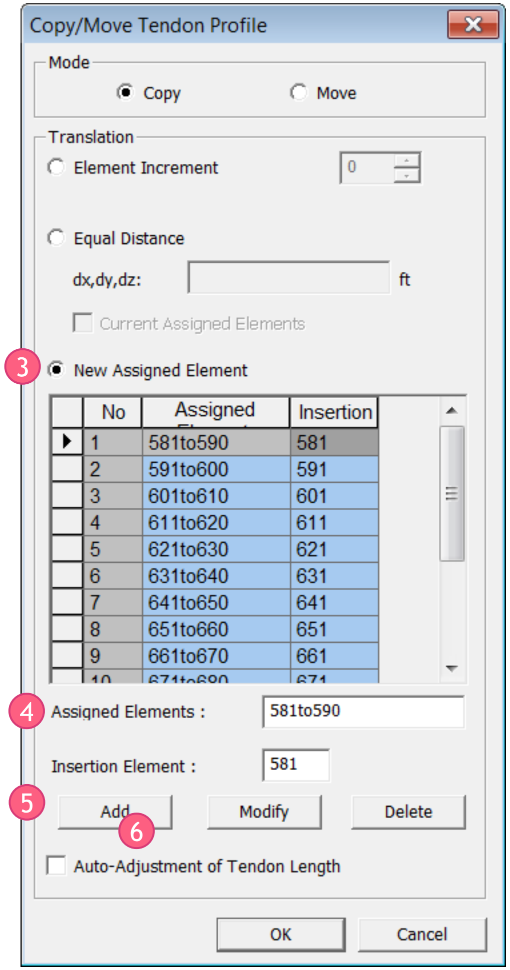
Load → Prestress Loads → Tendon Profile

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



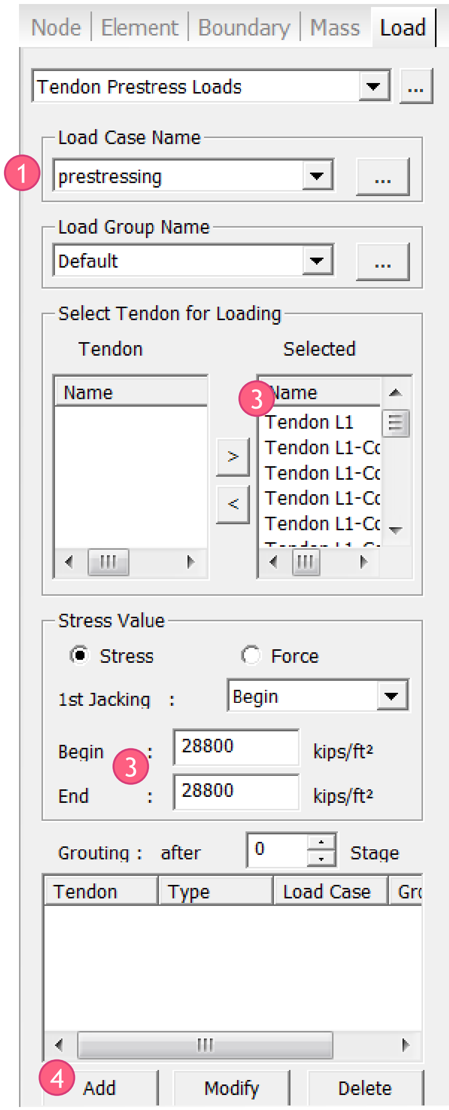
581to590	581
591to600	591
601to610	601
611to620	611
621to630	621
631to640	631
641to650	641
651to660	651
661to670	661
671to680	671
681to690	681
691to700	691
701to710	701
711to720	711

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

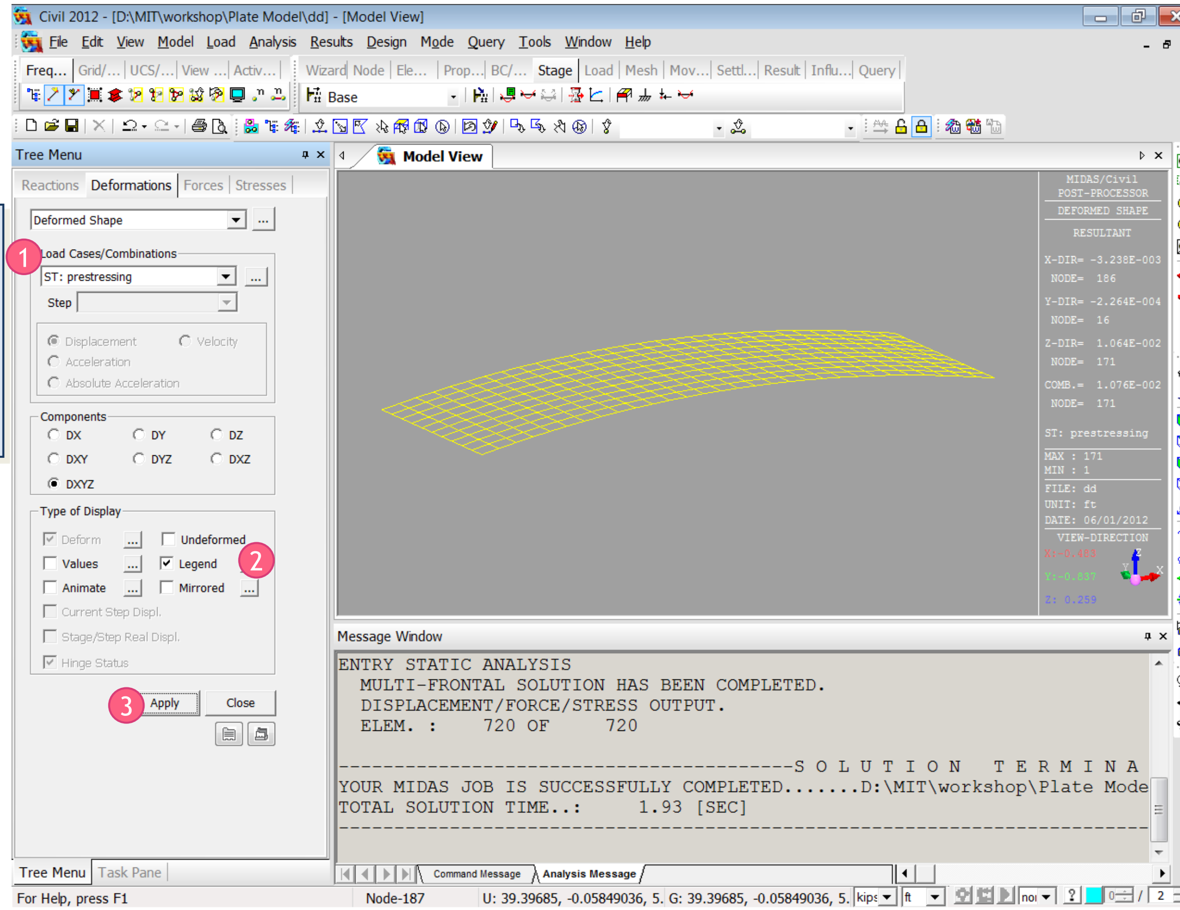
Analysis -> Perform Analysis



Results

Results → Deformations

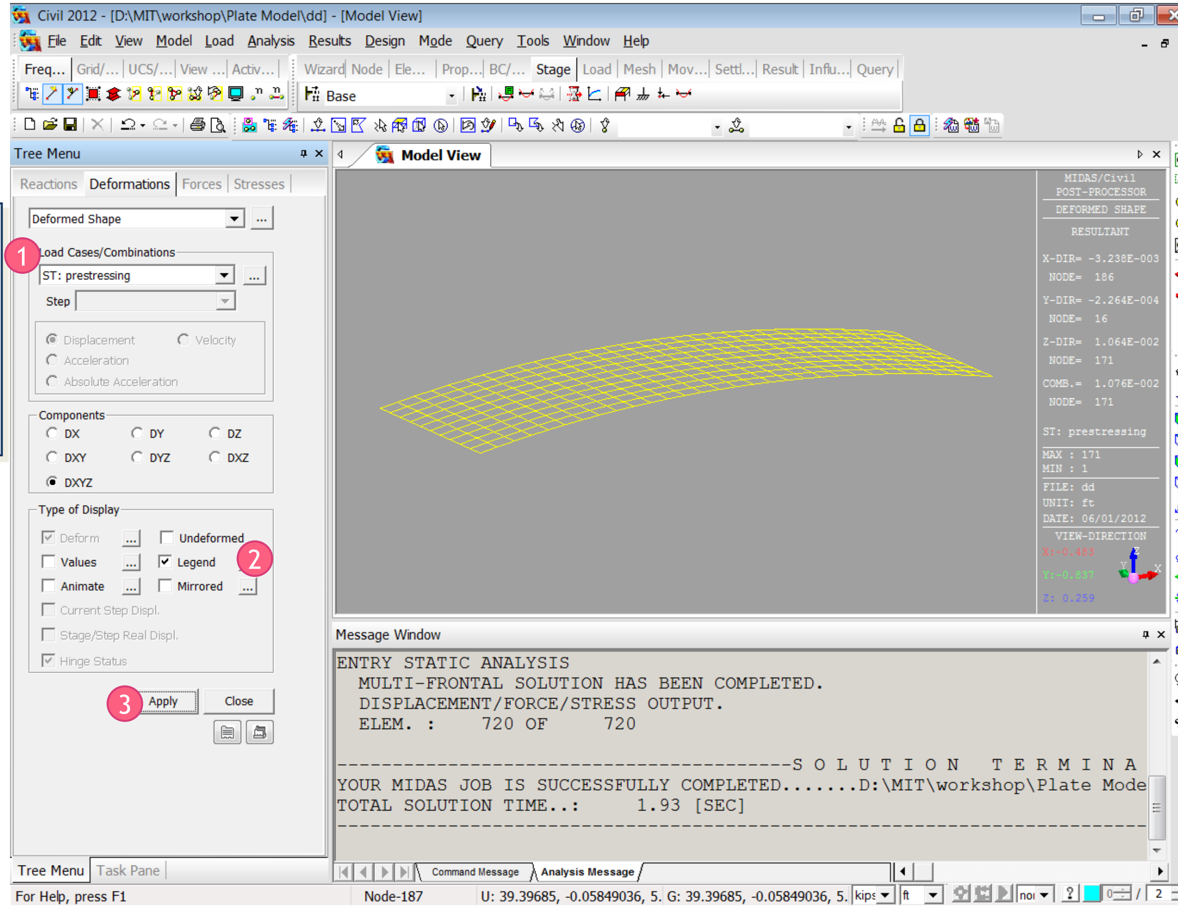
1. Load case → prestressing
2. Check on Legend
3. Click → Apply



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
646 852 9284